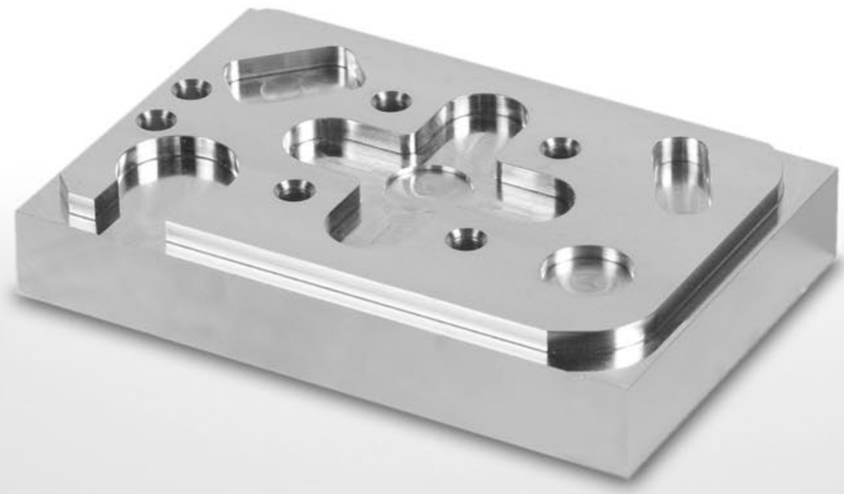
HEIDENHAIN



HIT Workbook
Fundamentals of Milling
HEIDENHAIN
Conversational Programming

TNC 320
TNC 620
iTNC 530

English (en)
V2
6/2012

Authors:

Gerhard Müller (CNC-Verlag)
Joachim Siebler (CNC-Verlag)

The HEIDENHAIN learning concept is suited for the following controls:

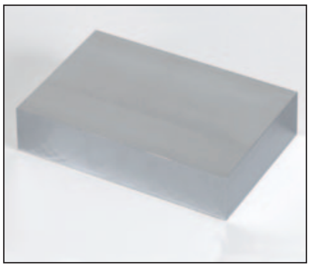
- iTNC 530
- TNC 620
- TNC 320
- TNC 430
- TNC 426

Contents

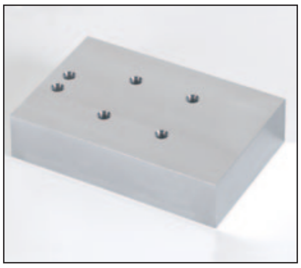
The three components of the HEIDENHAIN learning concept

HIT – The learning concept for HEIDENHAIN controls

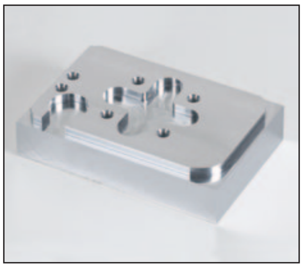
The HIT software, programming station and HIT workbooks are the three components of a new learning concept. The goal is to be able to write the NC program for the "pattern plate" workpiece using the HEIDENHAIN programming station.
Your "pattern plate" workpiece will accompany you throughout the course, growing at each stage.



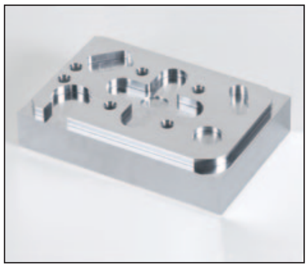
Initial state



... with drill holes



... and with contours



... and with cycles

Workpiece blank → **Finished part**

The HIT workbook and the HIT software assist you.

The HIT workbook contains assignments to be completed with the HIT software. The HIT software helps you learn the fundamentals of CNC technology, how to operate the HEIDENHAIN programming station, and how to create programs with the HEIDENHAIN conversational programming language.

The green fields in the workbook refer to the corresponding chapters of the HIT software, for example "Contour programming."

5. Contour programming

Circles CC/C

In order to program circular contours with C and CC, first work your way through the corresponding chapter. There you will learn how to program using the C and CC commands. The HIT workbook presents you with questions and additional assignments regarding this topic, for direct solving with the programming station.

Document your results in the workbook.

There are tests at the ends of chapters 4, 5 and 8 to determine how well you have learned the material.

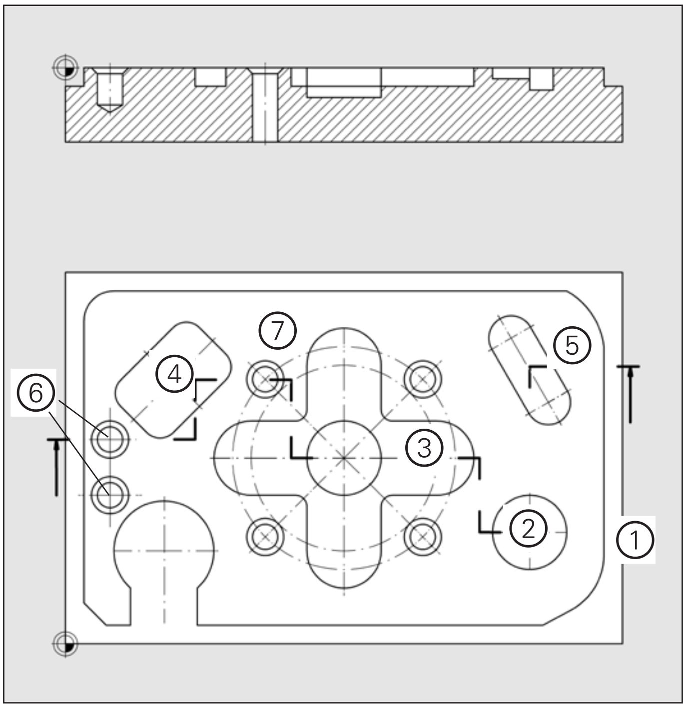
We wish you much joy and success with the HEIDENHAIN learning concept.

The Authors

Goal:
Once you have completed the workbook, you can competently program the "pattern plate" workpiece.

Write the following terms next to the appropriate number:

- Circular pocket
- Rectangular pocket
- Slot
- Pitch circle
- Holes
- Contour pocket
- Contour



1: ______________________________

2: ______________________________

3: ______________________________

4: ______________________________

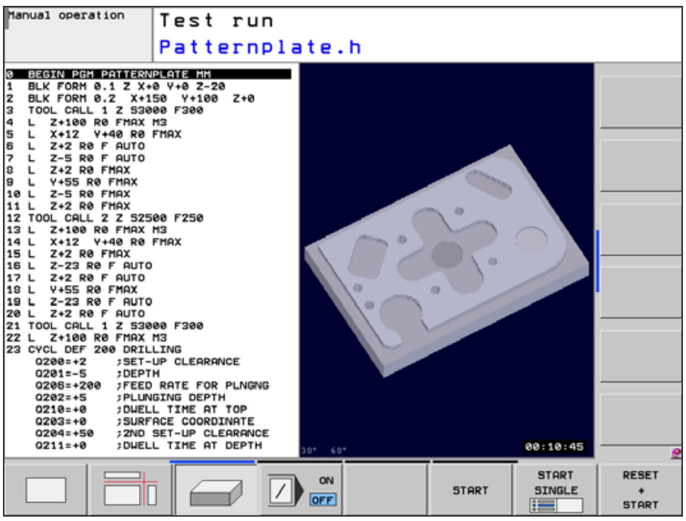
5: ______________________________

6: ______________________________

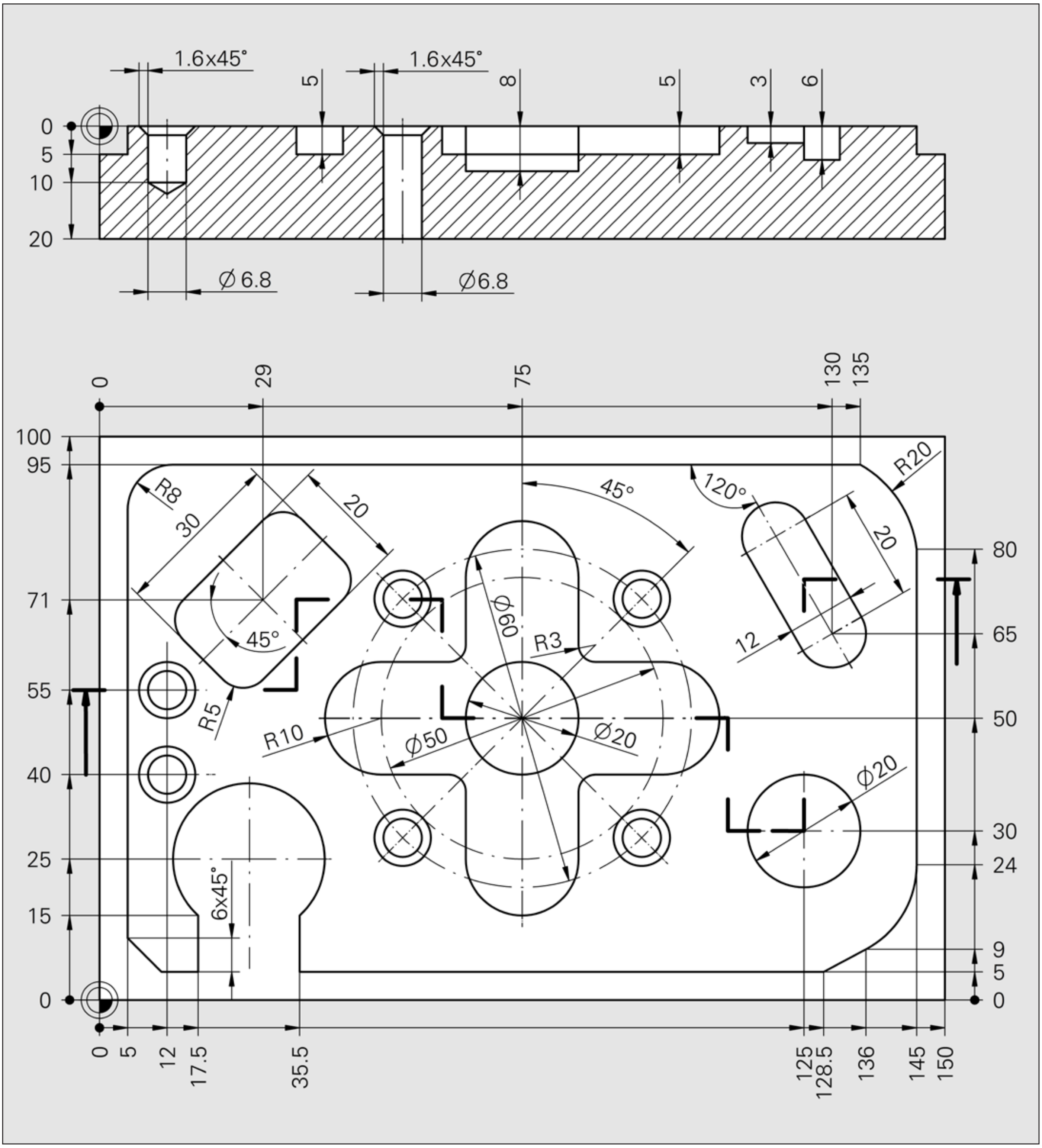
7: ______________________________

Here you can see the finished program for the "pattern plate" workpiece.

The graphic of the workpiece is displayed in the **Test Run** operating mode.

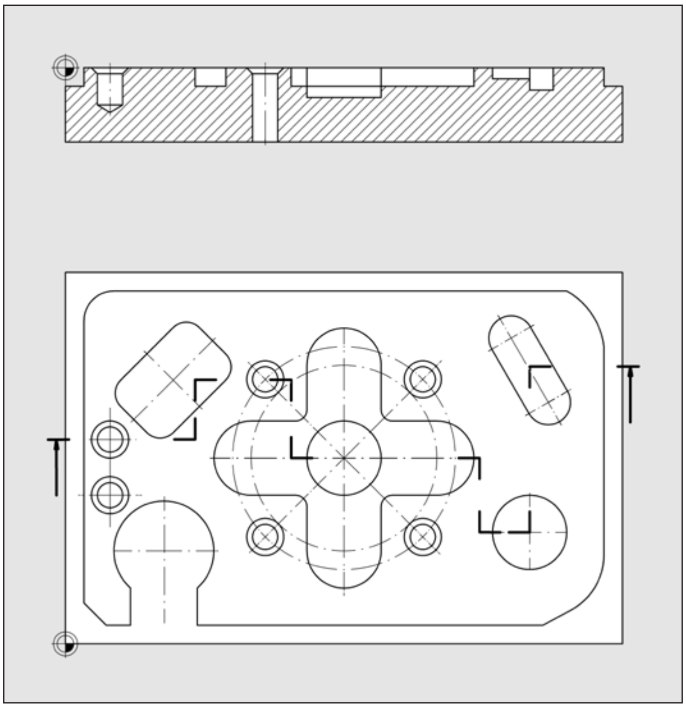


"Pattern plate" workpiece



Determine which tools you need to produce the "pattern plate" workpiece.
Use the tools that you actually have in your workshop.

These will be entered in the control later.



Tool table

1: ______________________________

2: ______________________________

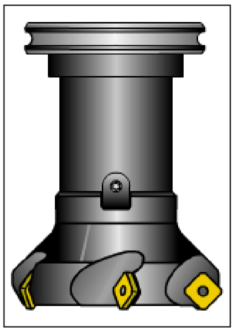
3: ______________________________

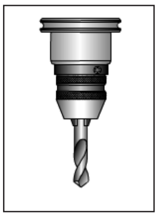
4: ______________________________

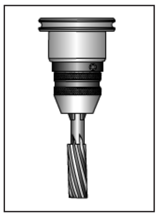
5: ______________________________

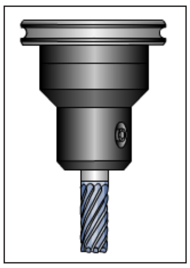
6: ______________________________

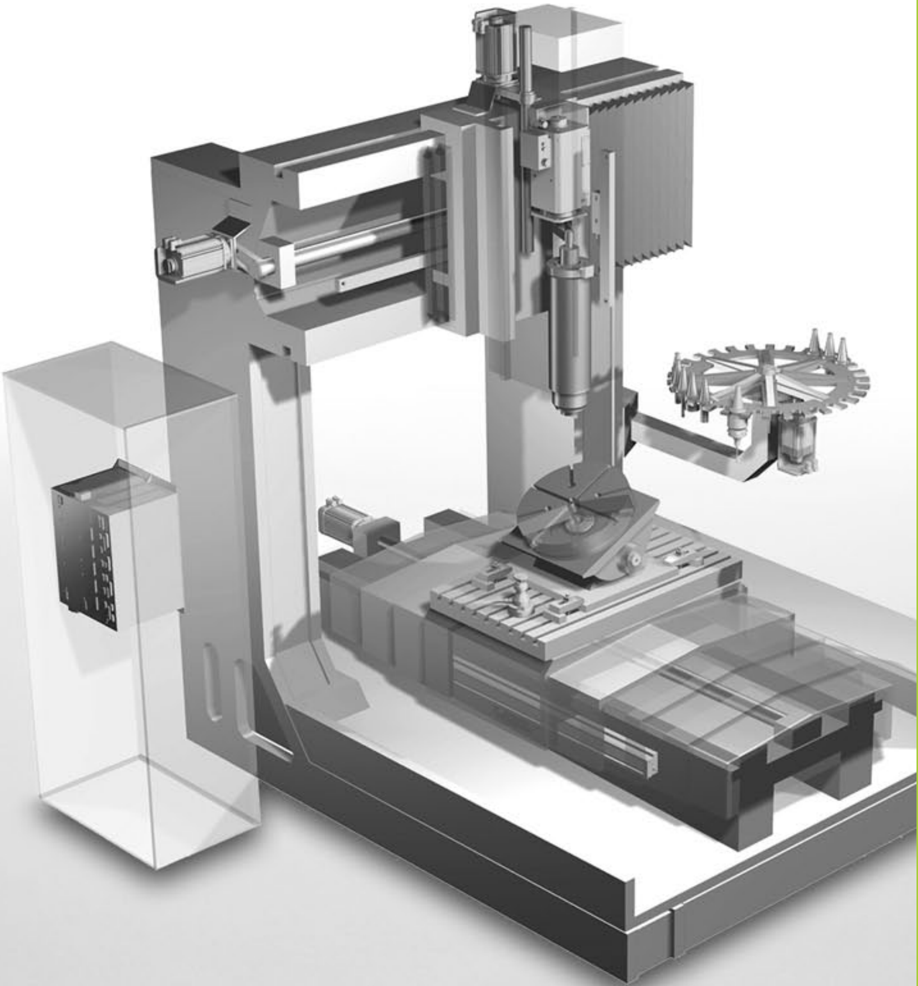
7: ______________________________











1

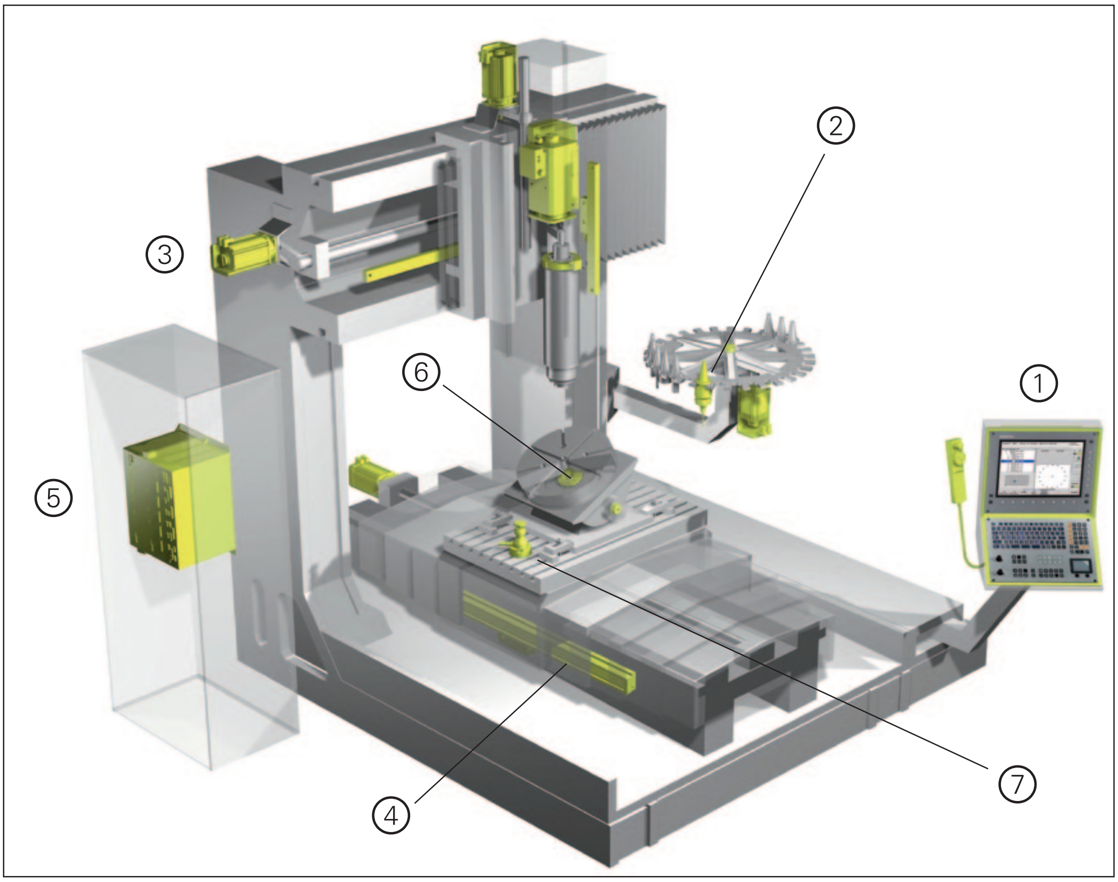
The CNC milling machine

1 The CNC milling machine

1.1 Setup

1. The CNC milling machine

Setup



Name the green elements of the machine tool.

1: ____

2: ____

3: ____

4: ____

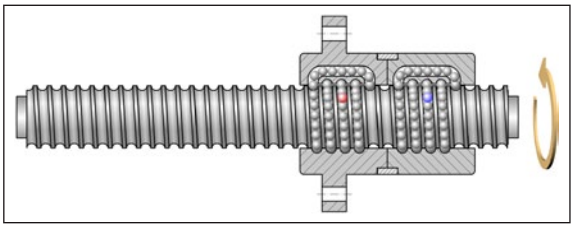
5: ____

6: ____

7: ____

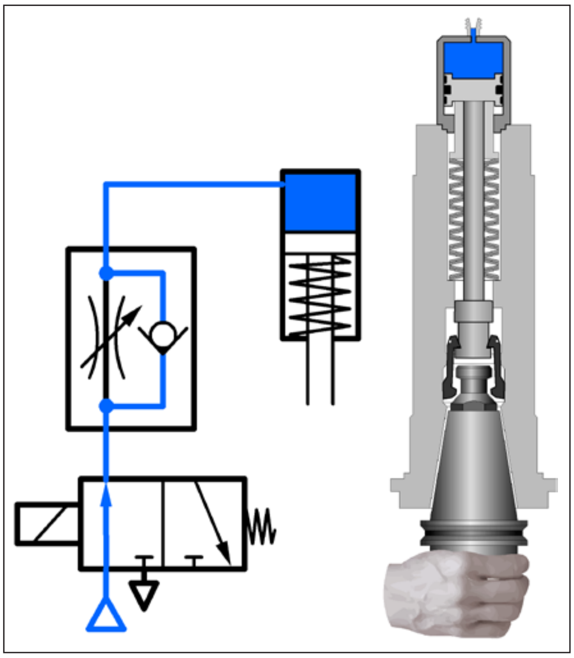
Mechanical components
In this section you will learn about the typical components of a CNC machine.

Recirculating ball screw
What is the function of the recirculating ball screw? Why are the two ball screw nuts tensioned opposite to each other?

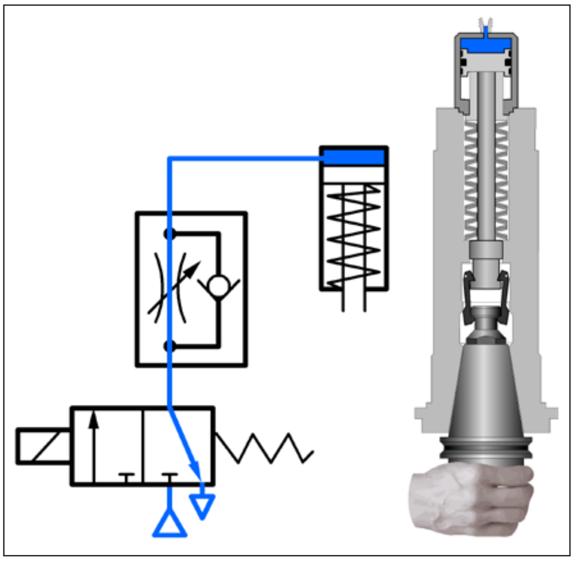


Tool holder

Explain what the diaphragm spring assembly does.



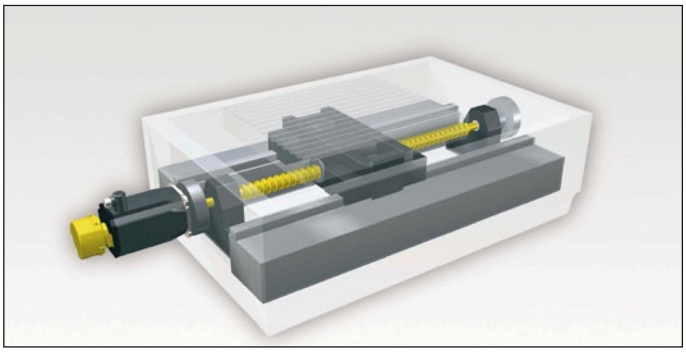
What is the function of the throttle-check valve?



Path measurement systems

Two methods are available for measuring the path traversed.

Measuring with rotary encoder and ball screw



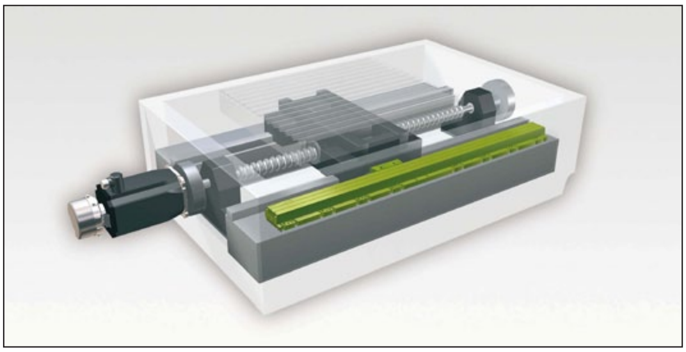
Where is the rotary encoder attached?

State the values needed in order to determine the position.

What is the purpose of a rotary encoder?

What is the disadvantage of measuring the path using this method?

Measuring with linear encoder

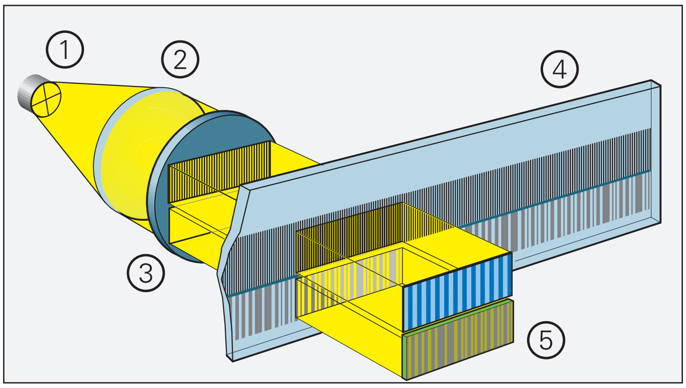


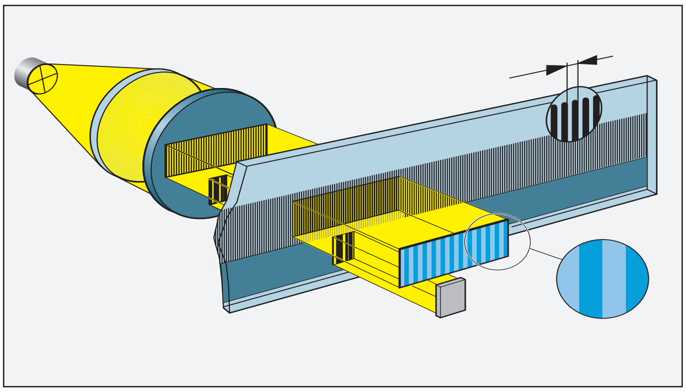
Where is the linear encoder mounted?

What is the advantage of this method?

Absolute and incremental path measurement

Identify the two types of systems.





Identify elements 1 through 5.

1:

2:

3:

4:

5:

What is the functional principle behind incremental path measurement?

Explain how absolute path measurement works.

Why do incremental path measurement systems need reference marks?

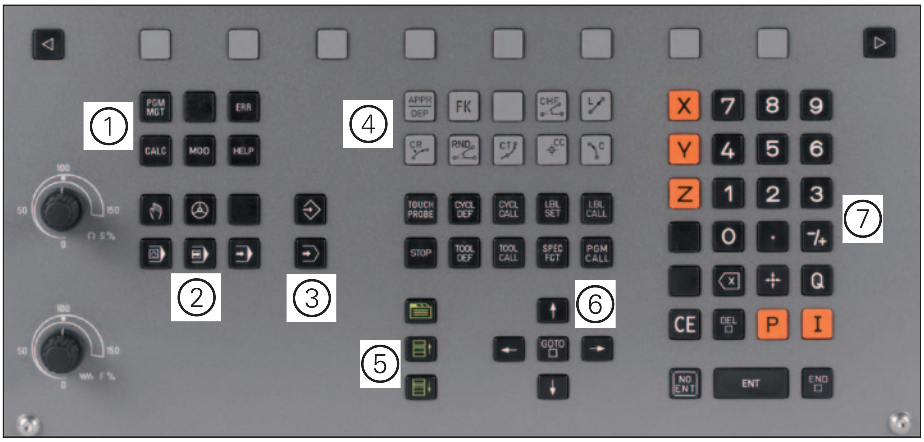
1.2 Keyboard overview

Keyboard of the TNC 620 / TNC 320

1. The CNC milling machine

Keyboard overview

Name the groups of elements of the TNC 620 / TNC 320 programming station.



1: ______________________

2: ______________________

3: ______________________

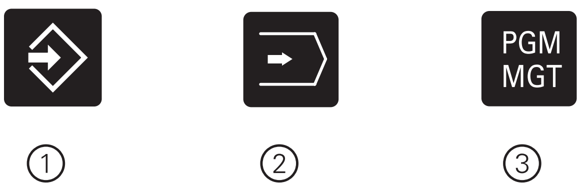
4: ______________________

5: ______________________

6: ______________________

7: ______________________

The following keys are required frequently.
What are their names?



1: ______________________

2: ______________________

3: ______________________

Keyboard of the iTNC 530

Programming and operation of the TNC 320 / TNC 620 is similar to that of the iTNC 530.
The iTNC 530 additionally features a touchpad as well as an alpha-numeric keyboard.

1: Alpha-numeric keyboard for entering texts and file names, as well as for programming in ISO

2: Touchpad

1.3 Screen layout

1. The CNC milling machine

Screen layout

Identify the various areas.

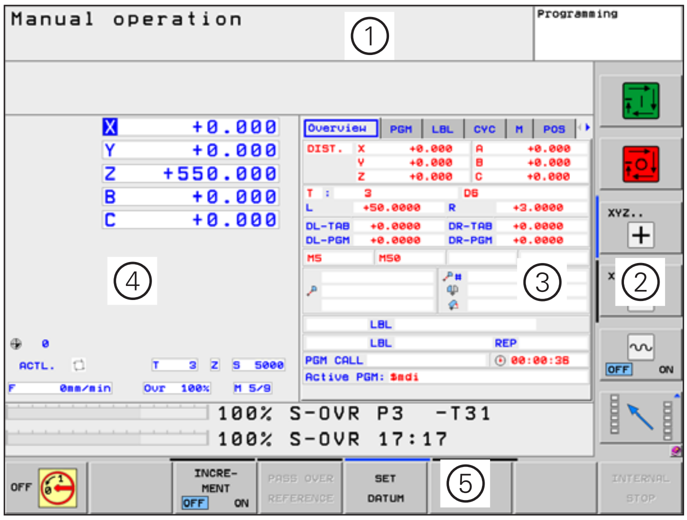
1: ____________________

2: ____________________

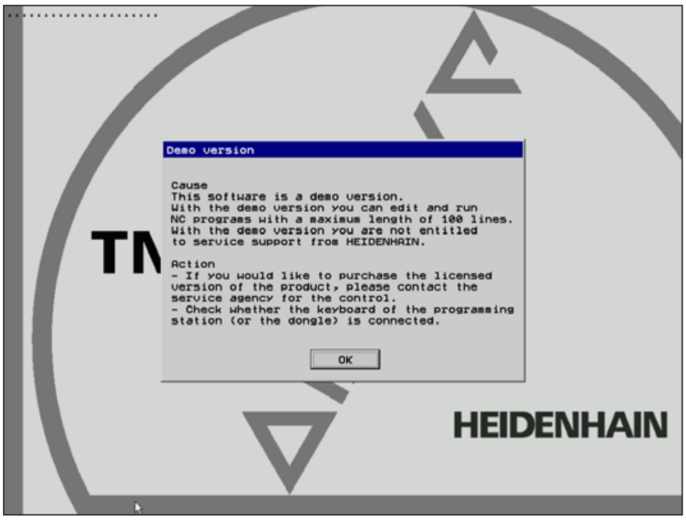
3: ____________________

4: ____________________

5: ____________________



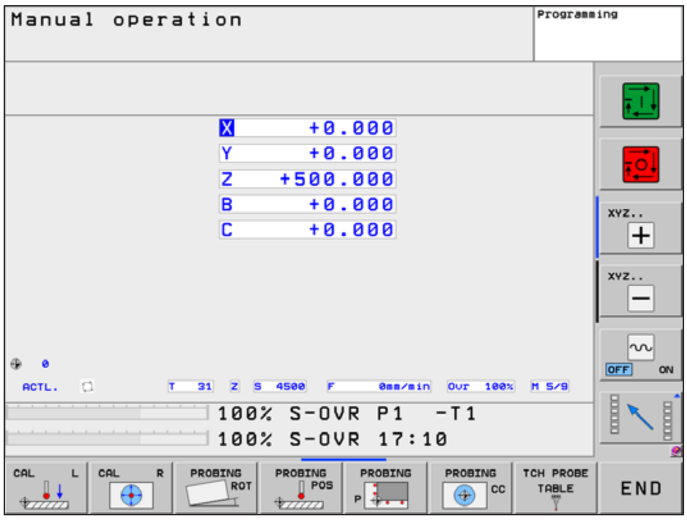
Start the programming station.
The following screen appears:



If you are using the demo version, click OK in the pop-up window.

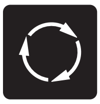
Press the CE key several times.
The CE key (Clear Entry) is used to acknowledge dialogs and error messages.

The following screen appears:



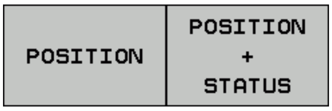
If you have not installed the programming station yet, you will find assistance in the installation help of the HEIDENHAIN Interactive Training program.

Press the screen layout key.



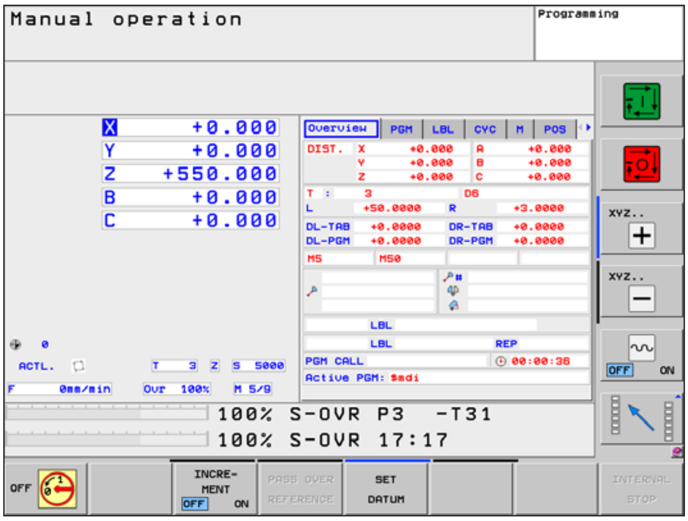
You can choose among several different views using the screen layout key.

They can be selected from the soft-key row.



Press the POSITION+STATUS soft key.

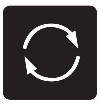
The following screen appears:



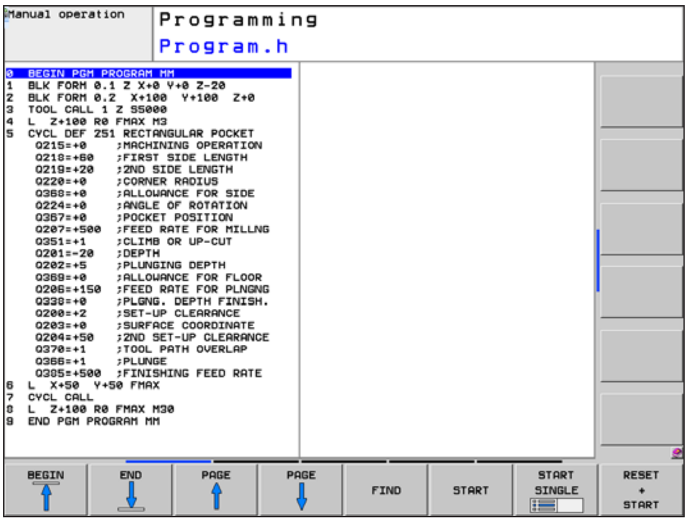
Press the screen layout key to switch to a different view.

You must either make a choice, or cancel with the END soft key to return to the previous functions.

With the operating-mode switchover key you can switch between the active machining mode and the active programming mode.



You will learn more about the operating modes in the next chapter.

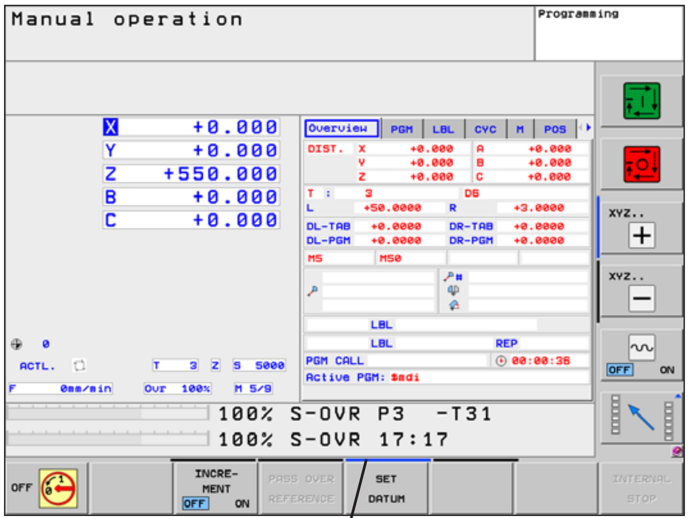


Shutting down the programming station

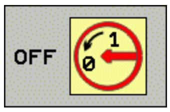
In order to avoid losing data when shutting down the programming station, you must exit it correctly.

▶ Press the **`Manual Operation`** operating mode key: the TNC switches to that mode.

▶ Shift the soft-key row until the OFF soft key for shutting down the system appears.

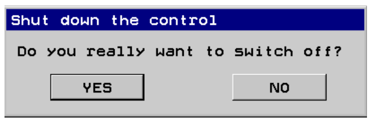


▶ Press the OFF soft key.



You can switch the soft key row here. You do this by clicking the thin bars, or pressing the F9 or F10 function keys.

▶ Answer the subsequent question in the pop-up window with YES.



Exiting the programming station inappropriately can lead to a loss of data.

1.4 Operating modes

1. The CNC milling machine

Operating modes

Identify the individual operating modes.

1: ____________________

2: ____________________

3: ____________________

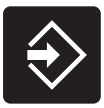
4: ____________________

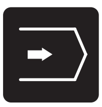
5: ____________________

6: ____________________

7: ____________________

What do you do in the following operating modes?





Grouping of operating modes

The operating modes are divided into two groups:

Identify them and explain their functions.

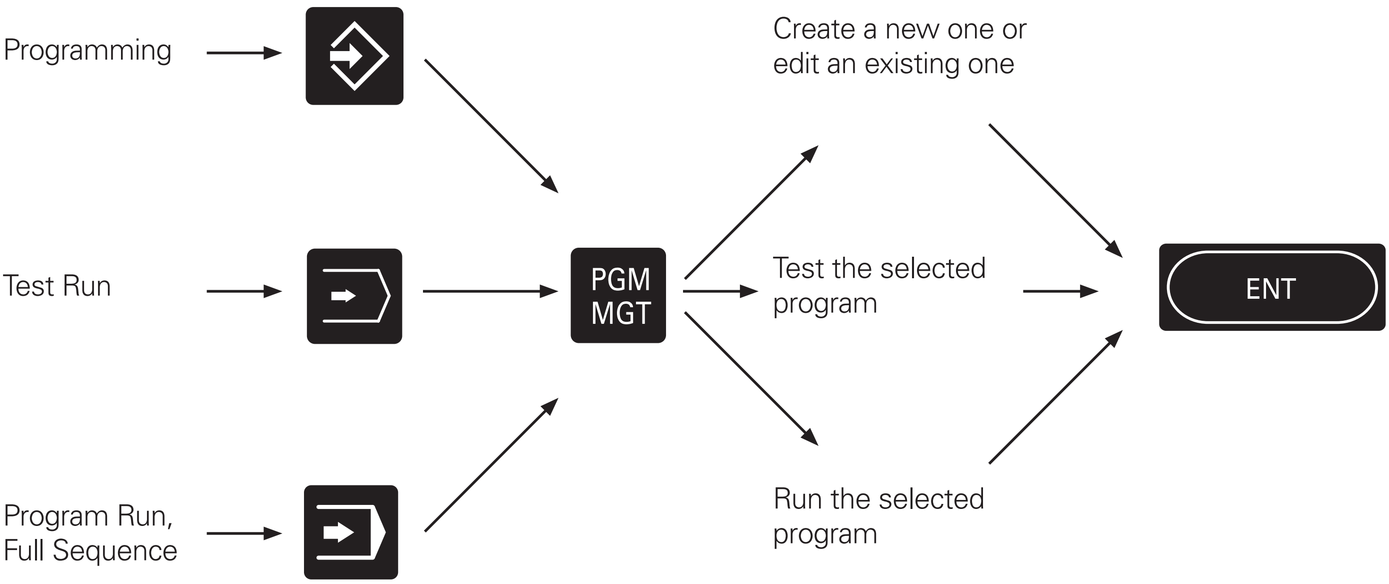
1: ______________________________

Functions: ______________________________

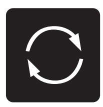
2: ______________________________

Functions: ______________________________

The file management must always be called after selecting the operating mode.



One mode of operation for machining and one for programming are always active. Switch between them via the operating-mode switchover key.



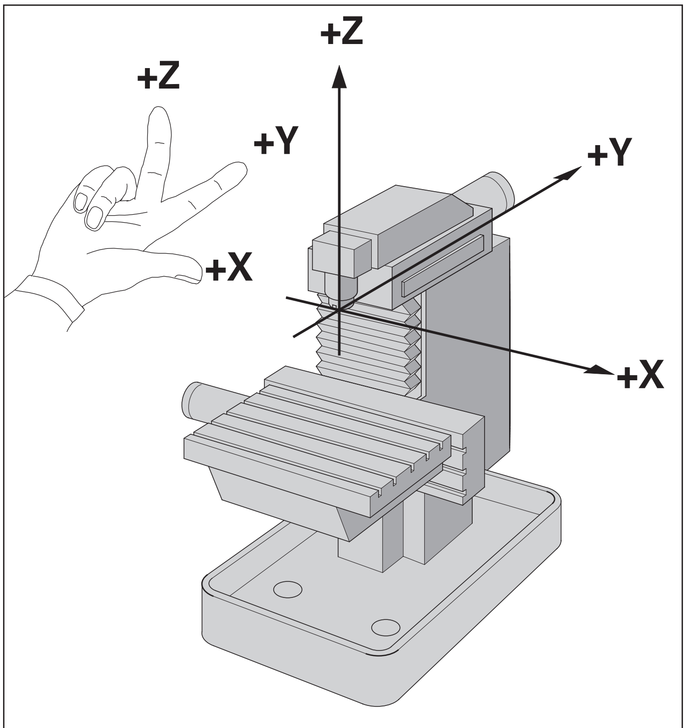
1.5 Moving the axes

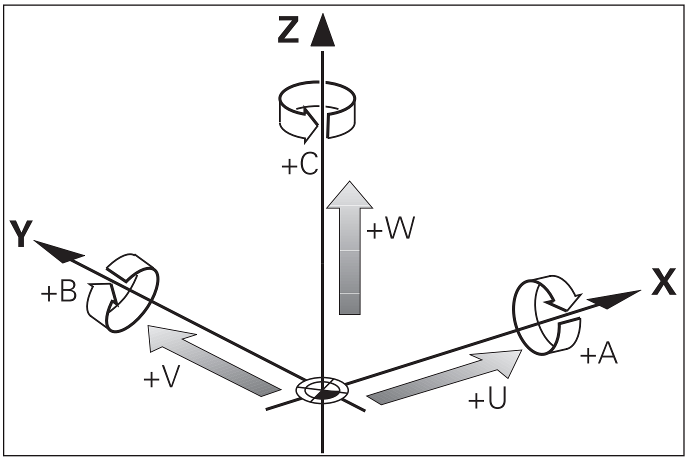
1. The CNC milling machine

Moving the axes

Axes as per ISO 841 (DIN 66217)

The following rules apply to the arrangement of the axes.





Principal axes	Rotary axes	Parallel axes
X	A	U
Y	B	V
Z	C	W

- The programmer assumes that only the tool moves, not the workpiece!
- Always write the programs as if the tool were moving.

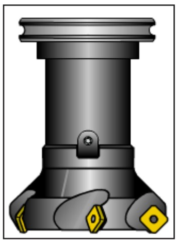
1.6 Tools

1.6.1 Tool types

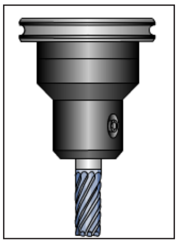
1. The CNC milling machine

Tools

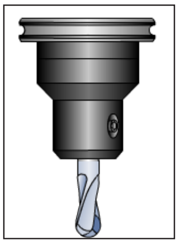
Identify the tools and state their purposes.



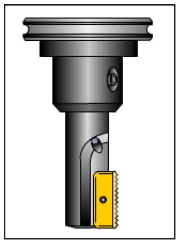
1



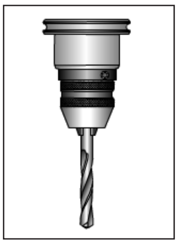
2



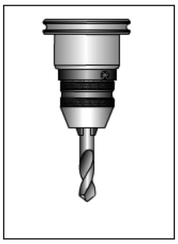
3



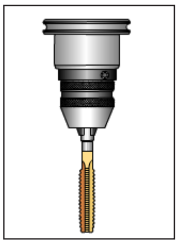
4



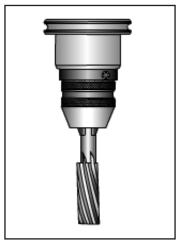
5



6



7



8

1:

2:

3:

4:

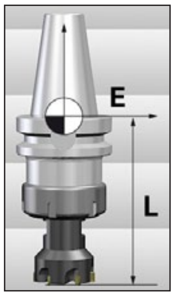
5:

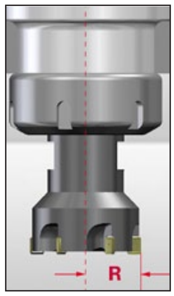
6:

7:

8:

1.6.2 Tool reference points



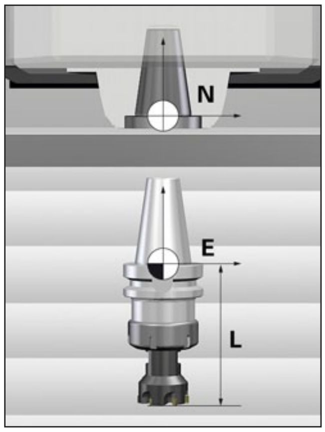


At least which two tool dimensions must be measured?

The geometry information of the tool is relative to which point?

Where is the geometry information stored?

Which points must coincide exactly?



Which point does the control use for motions in the spindle axis if the length 0 is entered in the tool memory?

The block L Z+2 FMAX of a program is run.
The tool length was accidentally entered 10 mm shorter than it really is.
What actual depth results?

There is a danger of collision if too short a length is entered for the tool!

The block L Z+2 FMAX of a program is run.
The tool length was accidentally entered 10 mm longer than it really is.
What actual depth results?

1.6.3 Tool measurement

State different possibilities for the measurement of tools.

How is the tool being measured in this example?

What are the benefits of contact-free measurement systems?

2

CNC fundamentals

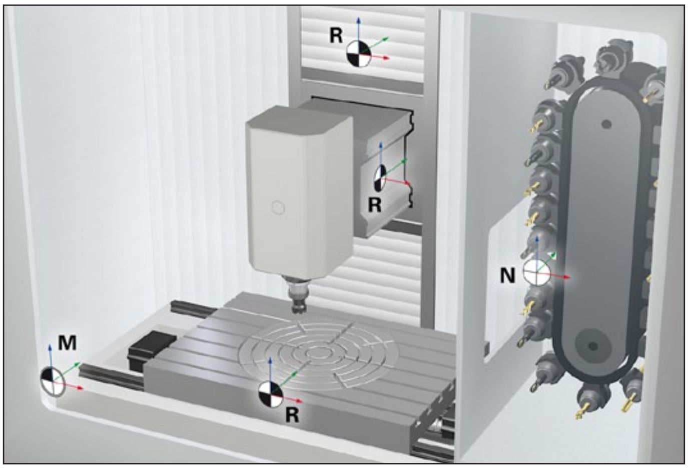
2 CNC fundamentals

2.1 Datums

2. CNC fundamentals

Datums

Workpiece datum, machine datum, reference point



Explain the following datums.

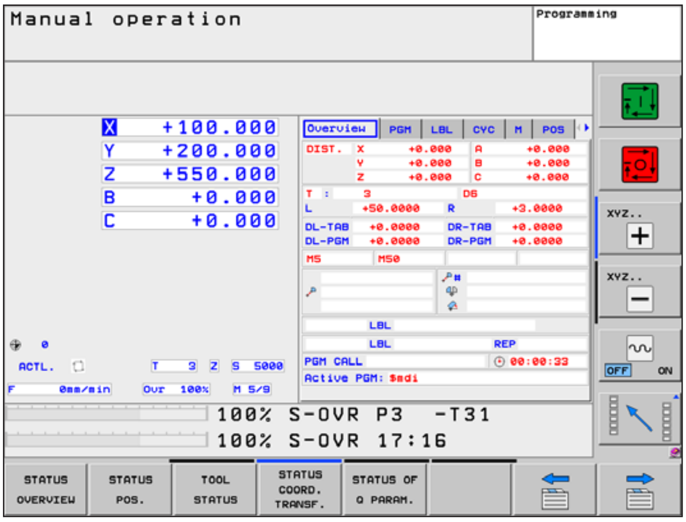
M:

W: (see middle figure on page 27)

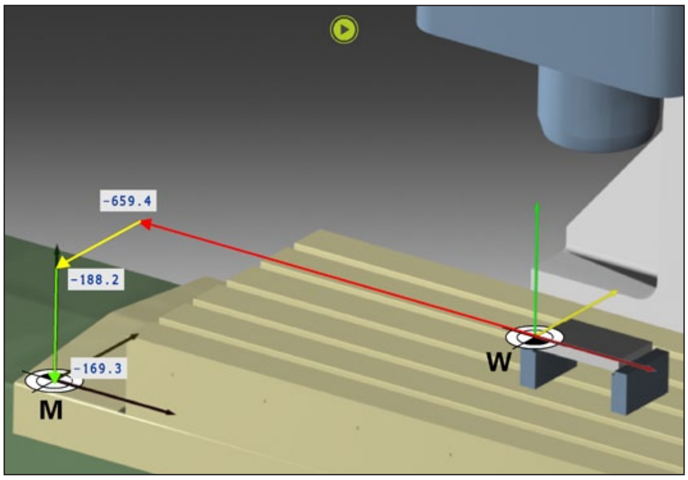
R:

In the figure, circle where the active preset is indicated.

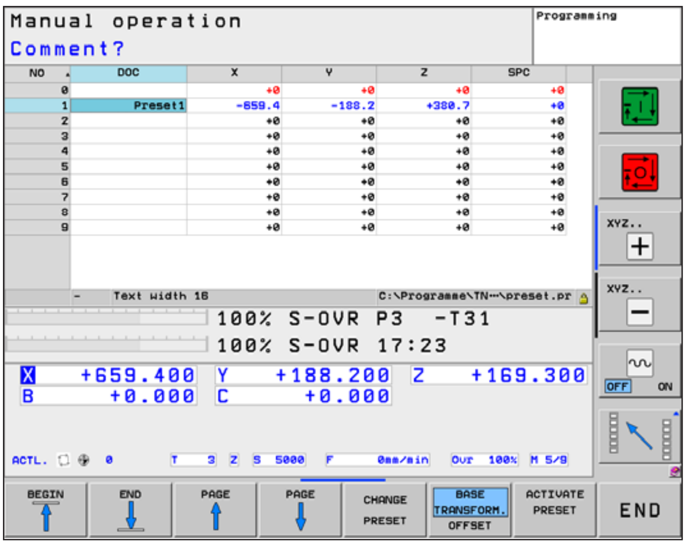
What point are the dimensions indicated there relative to?



Probing of the workpiece determines the distances from the workpiece datum W to the machine datum M.



They are saved in the preset table.

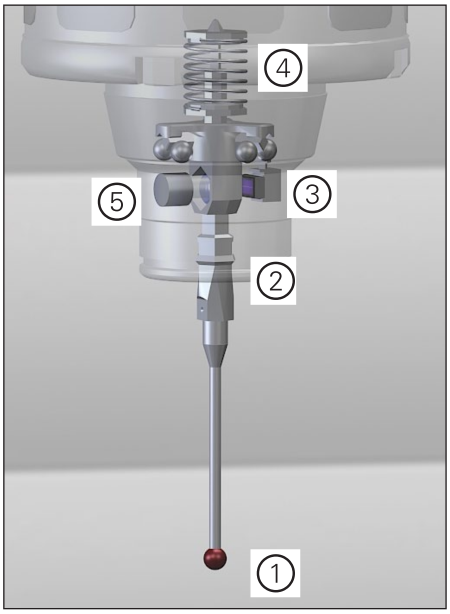


2.2 Touch probes

2. CNC fundamentals

3-D touch probe

Identify parts 1 through 5 and explain their functions.



1:

2:

3:

4:

5:

What initiates a trigger signal to the control?

State some tasks that can be accomplished with a 3-D touch probe.

What are the advantages of a 3-D touch probe over a dial indicator when setting up a workpiece?

2.3 Points on the workpiece

Hone your skills with Cartesian coordinates.
Work your way through the "Points on the workpiece" chapter.

2. CNC fundamentals

Points on the workpiece

2.4 Program layout

Identify the various elements of the program.

2. CNC fundamentals

Program layout

1	0 BEGIN PGM 112 MM
2	1 BLK FORM 0.1 Z X+0 Y+0 Z-20
	2 BLK FORM 0.2 X+120 Y+90 Z+0
3	3 TOOL CALL 5 Z S3200 F300
4	4 L X+30 Y+50 Z+1 R0 FMAX M3
	5 L Z-2 R0
	6 L X+120 R0
	7 L X+150 Y+100 Z+150 R0 FMAX
5	8 M30
	9 END PGM 112 MM

1: __________

2: __________

3: __________

4: __________

5: __________

Explain the following M commands.

M3: __________

M8: __________

M9: __________

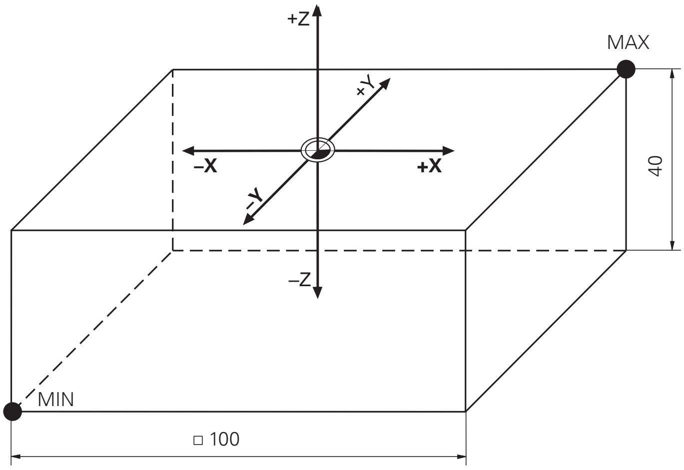
M30: __________

2.5 BLK form

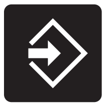
2. CNC fundamentals

BLK form

The workpiece blank in the drawing must be defined.
Start the programming station.



- Press the **`Programming`** operating mode key: the TNC switches to that mode.

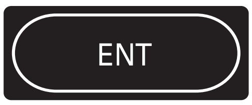


- Press the PGM MGT key: the TNC displays the file manager.

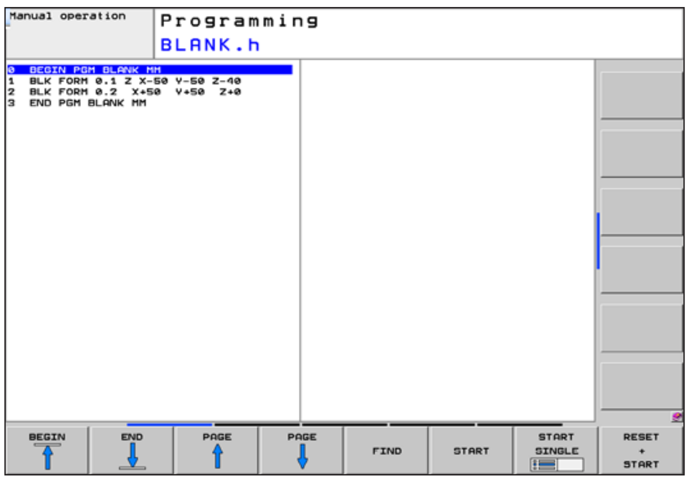
- Switch to the third soft-key row.
- Press the NEW FILE soft key.
- Enter Blank.H in the pop-up window.

The extension .H creates a conversational program.

- Confirm creation of the program with the ENT key.
- Select mm as the unit of measure.

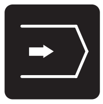


- Define the BLK form.*



*BLK=blank

▶ Press the **Test Run** operating mode key: the TNC switches to that mode.



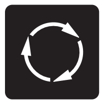
▶ Press the PGM MGT key: the TNC displays the file manager.

PGM MGT

▶ Press the LAST FILES soft key.
▶ Open the file Blank.H.

LAST FILES

Press the screen layout key and then use the soft-key row to choose from among the following views:



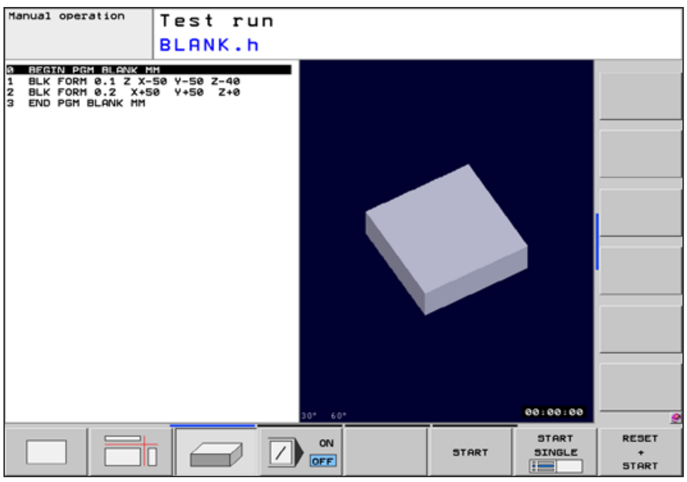
PROGRAM		PROGRAM + STATUS	PROGRAM + GRAPHICS	GRAPHICS		MODEL

▶ Press the PROGRAM+GRAPHICS soft key.

PROGRAM + GRAPHICS

The workpiece blank of Blank.H is displayed in the simulation graphics.

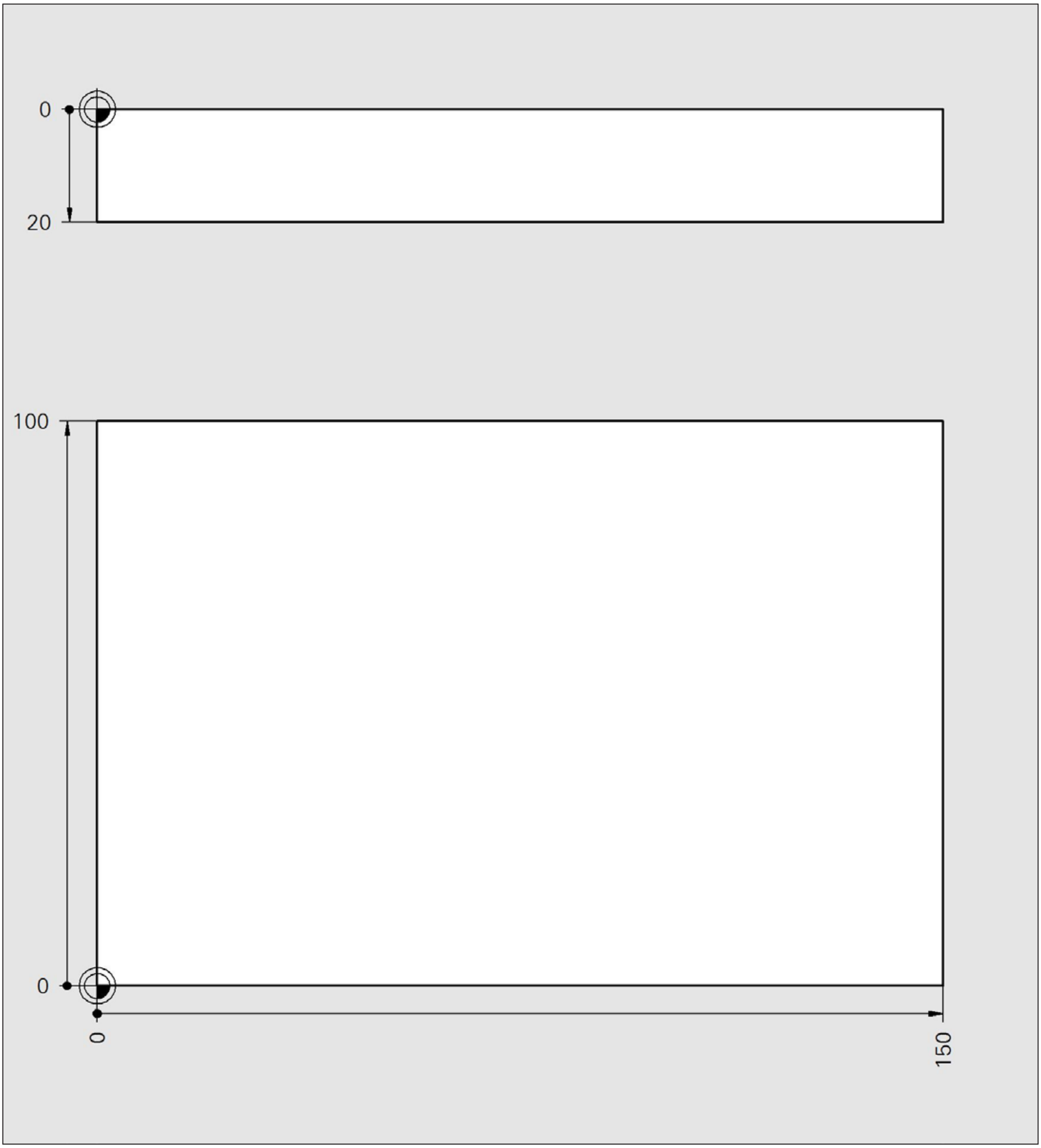
Use the soft keys above F1, F2 and F3 to choose between plan view, three-side view and 3-D view.



2. CNC fundamentals

Programming station

Start the programming station.
Create a program for the “pattern plate” workpiece and define the BLK form.



Exercises 2a, b, c, d

3

Technological fundamentals

3 Technological fundamentals

3. Technological fundamentals

Calculating the cutting data
Cutting data in theory

Production quality, safety and good production times all require optimal cutting data.

Which two pieces of cutting data must be determined?

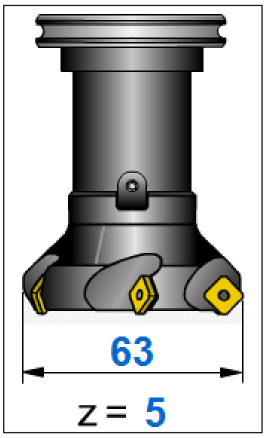
____

____

They are mostly determined from manufacturer catalogs.

Determine the shaft speed and feed rate for the milling head.
The cutting speed is 250 m/min, and the feed per tooth is 0.4 mm.

Calculations:



On the control the letter S indicates the spindle speed, and the letter F the feed rate.

Determine the necessary cutting data for the tools on page 8.
Use the available documentation.
Material: AlCuMgPbMn

Tool no.	Tool	Cutting material	v_c	f_z	z	S	F

Calculations:

Cutting data in practice

The values calculated for the feed rate and spindle speed on the previous pages are based on theoretical values.
In practice, however, these values depend on many factors, and must often be adjusted.

State the factors that must be considered.

These factors show how interwoven the different aspects are.
There are no obviously correct cutting data, just cutting data that is more or less correct.

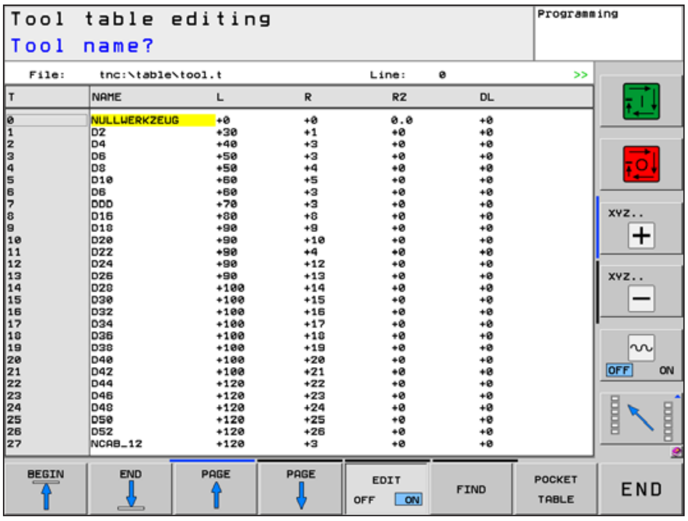
Being in contact with the tool manufacturer helps you to select the cutting data, but in the end you have to gather your own experiences. More experienced colleagues can help you with this.

Tool table

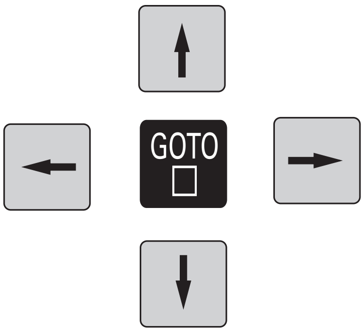
Enter the tools in the tool table.
Proceed as follows:

- Select the **`Manual Operation`** mode.
- Press the TOOL TABLE soft key.
- Press the EDIT ON soft key.

The following screen appears.



Press the arrow keys until you reach the fields to be edited, and enter the values.



> **DANGER OF COLLISION!**
> For real machining, enter the actual values for the length and radius!

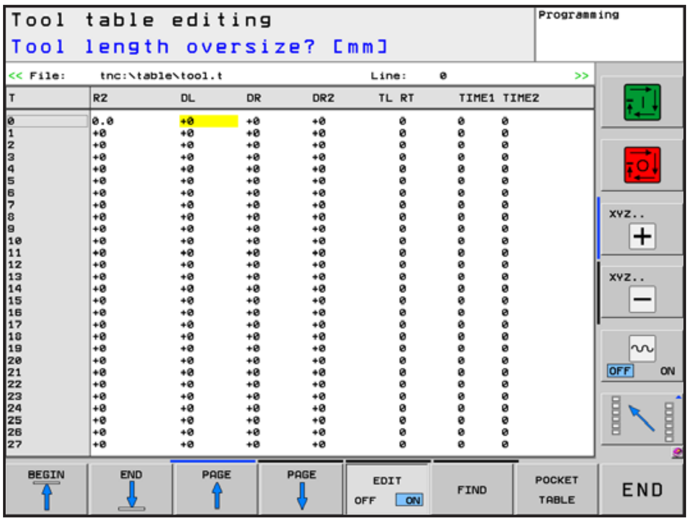
The control activates this data when a tool is called in the program via the TOOL CALL key.

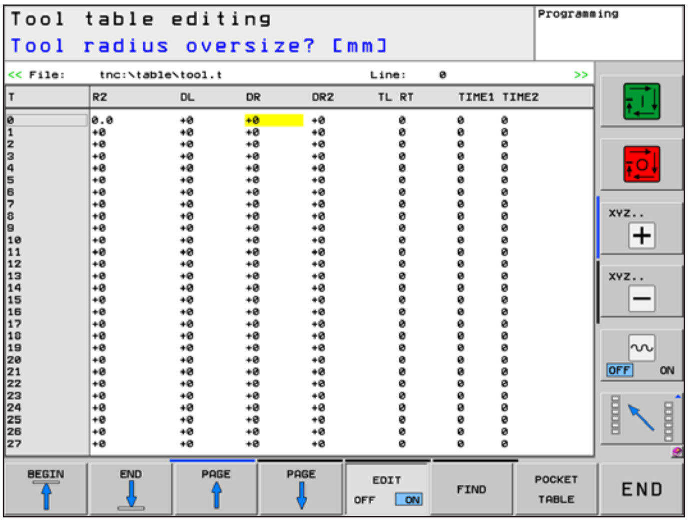
The entered values are stored by pressing the END key.
Always close the tool table with the END key.

Other tool data, along with the length and radius, can also be entered in the tool table.
However, only the length and radius are mandatory for production.

In these two windows an oversize for the tool length and tool radius is being edited.

This tool data will be explained later.



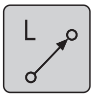


TNCguide has more information about the other tool data.

4

Linear motions

4 Linear motions



4.1 Positioning with FMAX

4. Linear motions

Positioning with FMAX

In this chapter you program the holes for the pattern plate.
You will learn the necessary path functions here.

Explain what FMAX is.

____

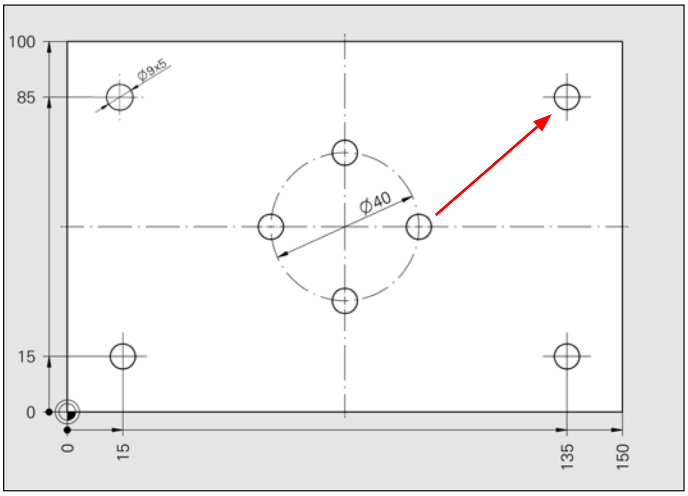
____

____

What is meant by "modally effective"?

____

____



For safety reasons, FMAX is only effective blockwise.

The pre-positioning to the next drill hole must be completed in this program excerpt.
Complete line 19.

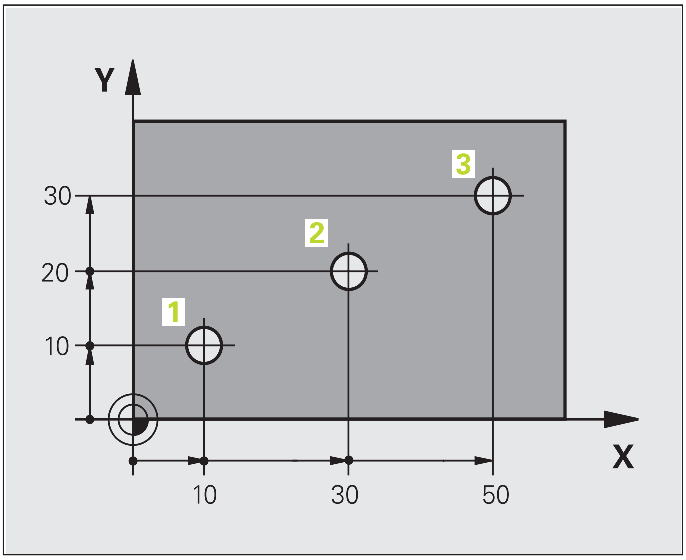
16 L X+95 Y+50 FMAX	Positioning
17 L Z-5 F250	Drilling
18 L Z+2 F250	Retraction
19	Positioning
20 L Z-5 F250	Drilling

4.2 Straight line (absolute)

4. Linear motions

Straight line (absolute)

- Absolute dimensioning refers to the workpiece datum.



Complete the table.

Hole 1	Hole 2	Hole 3
X =	X =	X =
Y =	Y =	Y =

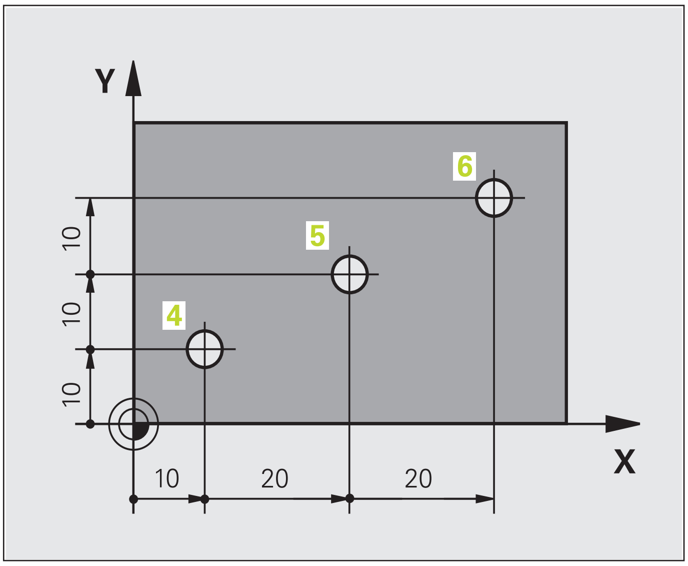
L X... Y...	... move to ...

4.3 Straight line (incremental)

4. Linear motions

Straight line (incremental)

- Incremental dimensioning refers to each momentary position of the tool.



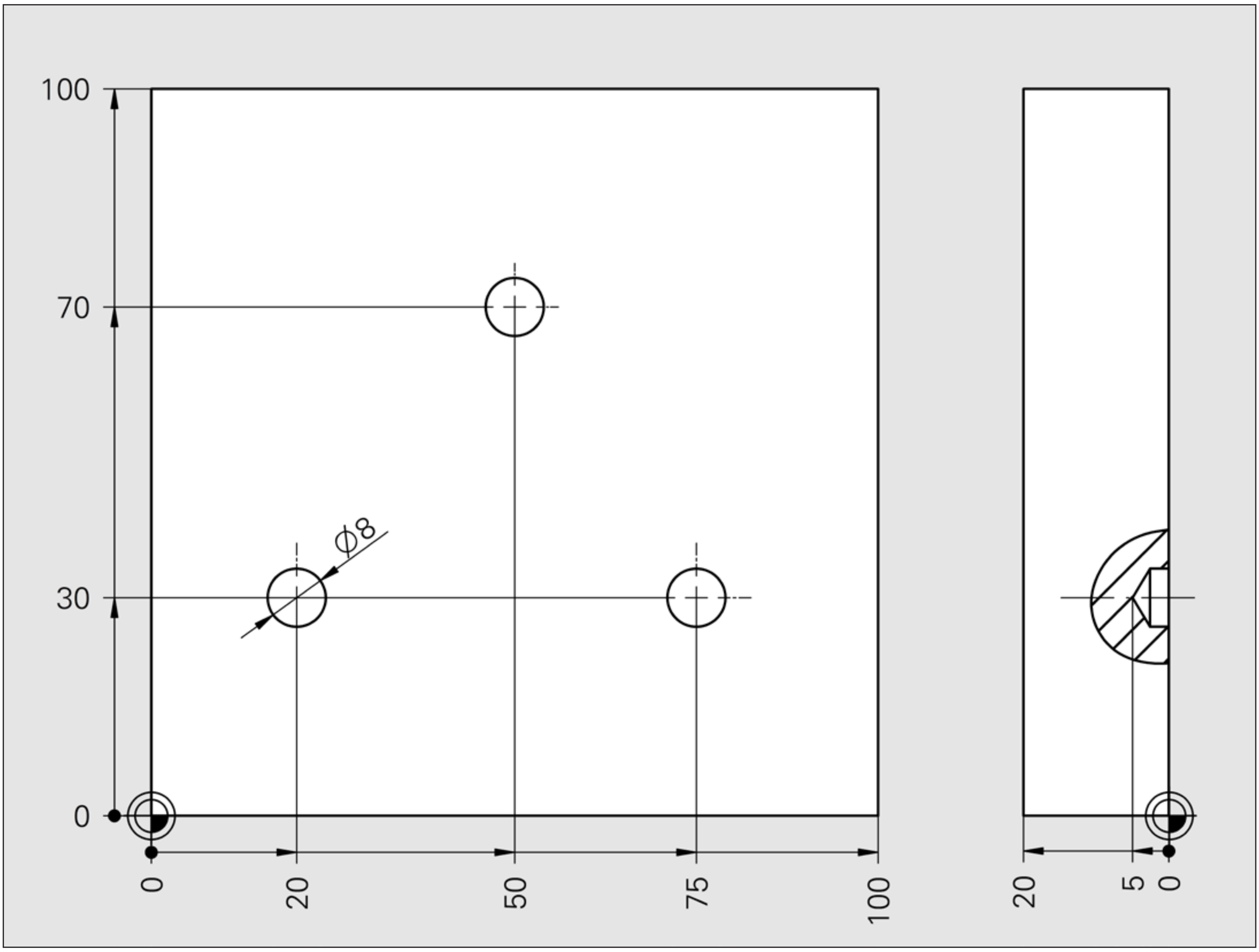
Complete the table.

Hole 4	Hole 5	Hole 6
X =	IX =	IX =
Y =	IY =	IY =

L IX... IY...	... move by ...

The I key on the HEIDENHAIN keyboard is used to switch between absolute and incremental programming.

Assignment: Holes
Program the following workpiece on your programming station.
Please observe the scheme on the next page.



Enter the required tool in the tool table.

Scheme: Holes

		Programming operating mode
PGM MGT	`BEGIN PGM ... MM`	Create program
	`BLK-FORM 0.1 ... X... Y... Z...` `BLK-FORM 0.2 X... Y... Z...`	Define workpiece blank
TOOL CALL	`TOOL CALL ... ... S...`	Activate tool
L	`L Z+100 R0 FMAX M3`	Move to clearance height
L	`L X... Y... R0 FMAX`	Approach hole position 1 (X=20, Y=30)
L	`L Z+... FMAX`	Move to set-up clearance
L	`L Z-... F250`	Drilling
L	`L Z+2... FMAX`	Retract the drill
L	`L X... Y... FMAX`	Move to next position
L	`L Z-...`	Drilling
L	`L Z+... FMAX`	Retract the drill
L	`L X... Y... FMAX`	Move to next position
L	`L Z-...`	Drilling
L	`L Z+100 R0 FMAX M30`	Return to clearance height
	`END PGM ... MM`	End of program

Create another program, this time with incremental positioning between the drill holes.

Exercises 4a, b, c

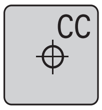
4.4 Polar coordinates: Straight line LP

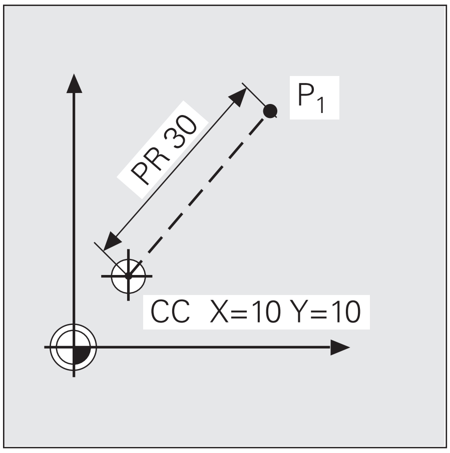
4. Linear motions

Polar coordinates

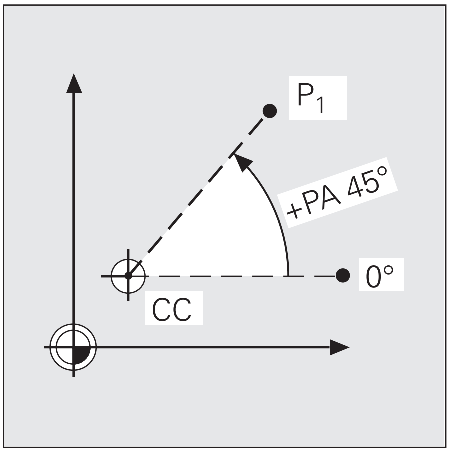
With polar coordinates you can define a position in terms of its distance PR and its angle PA relative to a previously defined pole.

The pole is specified with CC.





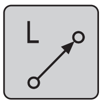
Polar coordinate radius PR:
Distance of position P_1 to the pole.



Polar coordinate angle PA:
Angle from the angle reference axis (= 0° line) to the line connecting the pole and P_1.

Inputting line 18:

- Select the path function.
- Press the polar key.
- Enter the polar radius **PR**.
- Enter the polar angle **PA**.



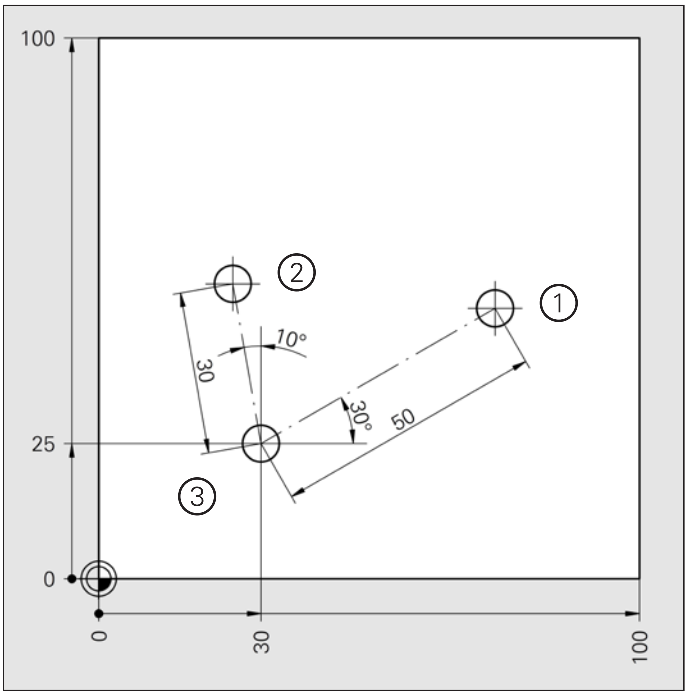
`16 L X+10 Y+10 FMAX`	Positioning
`17 CC`	CC assumes the current position as the pole
`18 LP PR+30 PA+45 FMAX`	Positioning with polar coordinates to P_1

The position of the pole can be entered directly in the CC block.

`17 CC X+10 Y+10`	Input of the pole directly in the CC block
`18 LP PR+30 PA+45 FMAX`	Positioning with polar coordinates to P_1

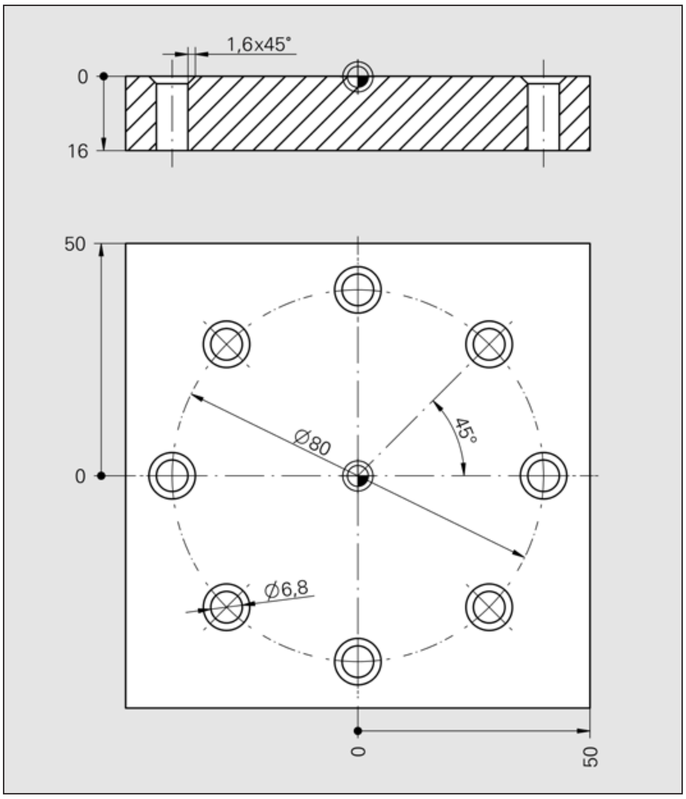
Programming the pole does not result in axis movement.

Complete lines 20 and 23.



`16 L X+30 Y+25 FMAX`	Positioning to hole 3
`17 CC`	Assume the actual position X+30 Y+25 as pole
`18 L Z-5 F250`	Drilling
`19 L Z+2 F250`	Retract
`20`	Positioning to hole 1
`21 L Z-5 F250`	Drilling
`22 L Z+2 F250`	Retract
`23`	Positioning to hole 2
`...`	

Write the program for the following workpiece on your programming station.



Exercise 4d

4. Linear motions

Summary

4. Linear motions

Writing an NC program

In the HEIDENHAIN Interactive Training program, enhance your knowledge with the Summary and Writing an NC Program sections.
These sections prepare you for the first test.

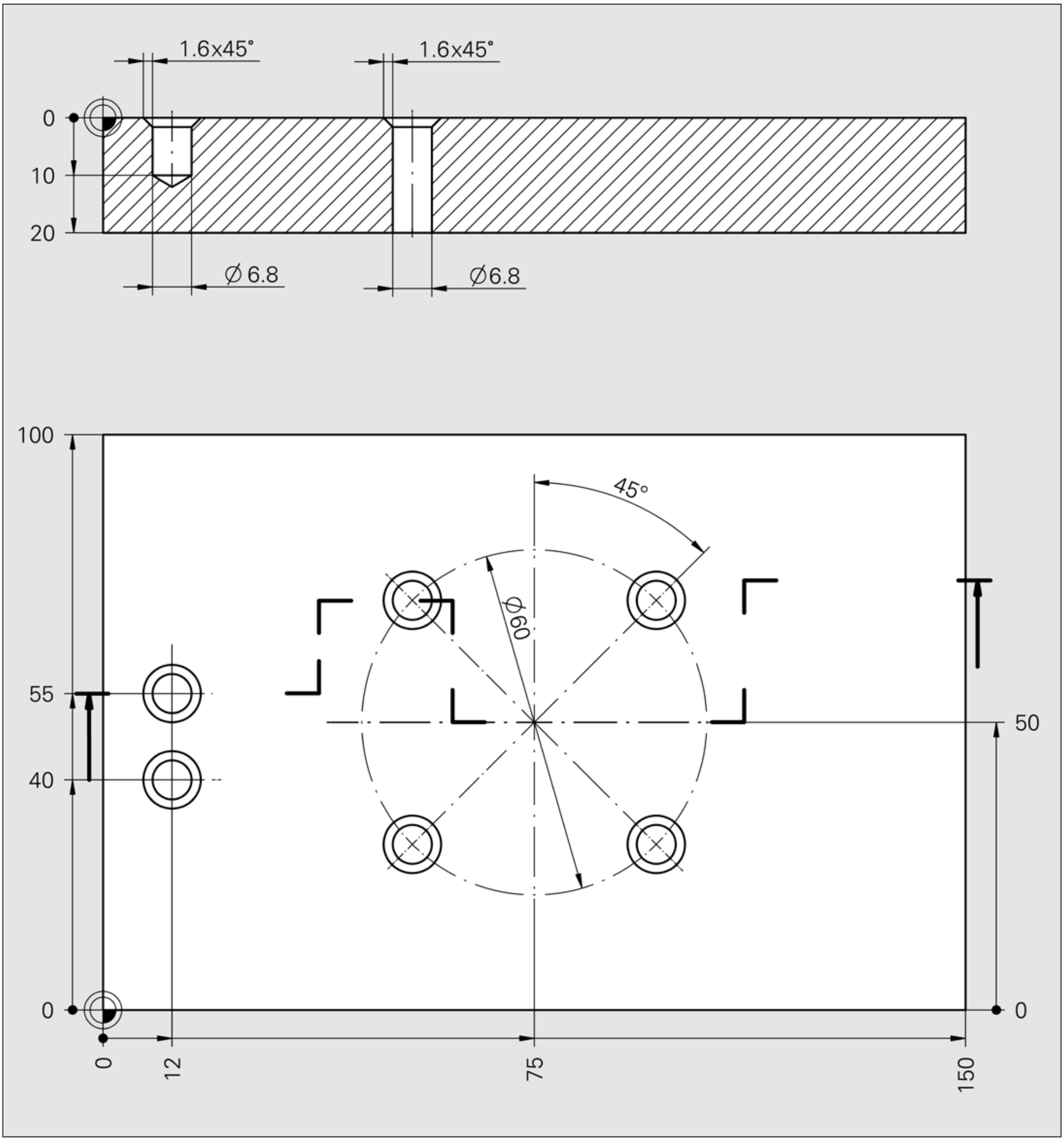
4. Linear motions

Test 1

Complete the first test and print it out.
Paste it in here for your documentation.
Paste it over the informational text on this page.
You can repeat the test as many times as you want until you are satisfied with the result.

Programming station

Start the programming station.
Add the drill holes to your program for the "pattern plate" workpiece.



5

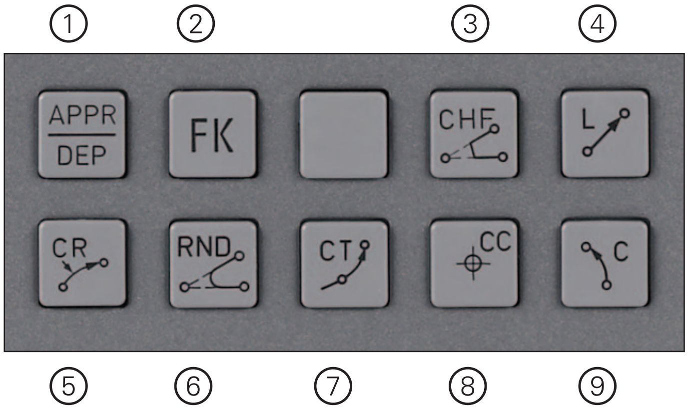
Contour programming

5 Contour programming

Contours consist of circular and straight elements. Programming dialogs are initiated with the gray dialog keys.

Identify the individual functions.

(See HEIDENHAIN Interactive Training, chapter 1, Keyboard overview)



1: __________

2: __________

3: __________

4: __________

5: __________

6: __________

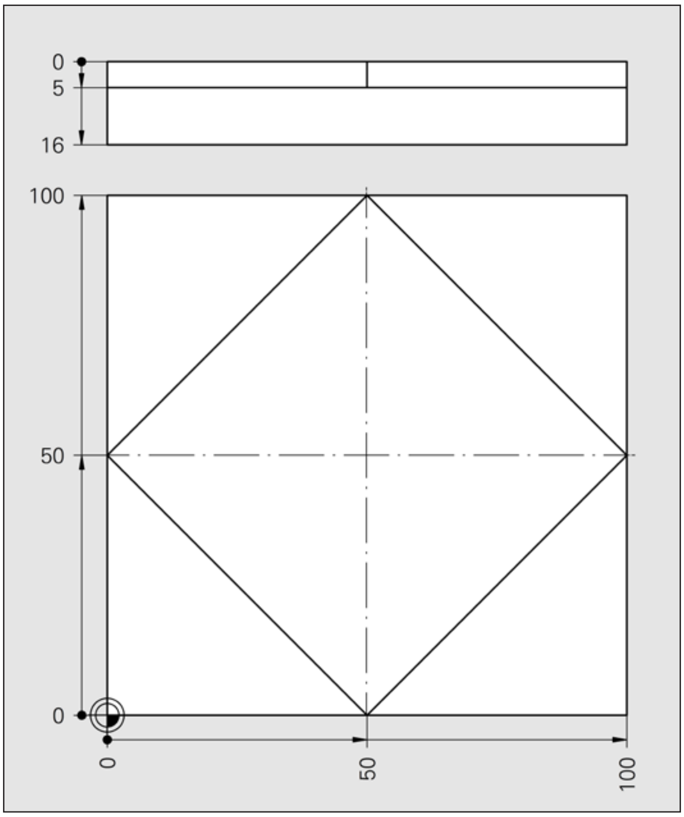
7: __________

8: __________

9: __________

5.1 Radius compensation

Write the program for the following contour.
Use a cutter with D = 25 mm.
The starting point for the contour is X0, Y+50.
(Name: Diamond.H)



Simulate the program. What do you see?

____

5. Contour programming

Radius compensation

Improve your program by using radius compensation.

Which point of the cutter is programmed with R0?

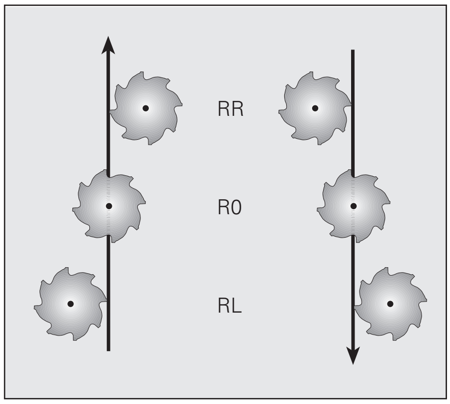
What does radius compensation do?

Explain RL, RR and R0.

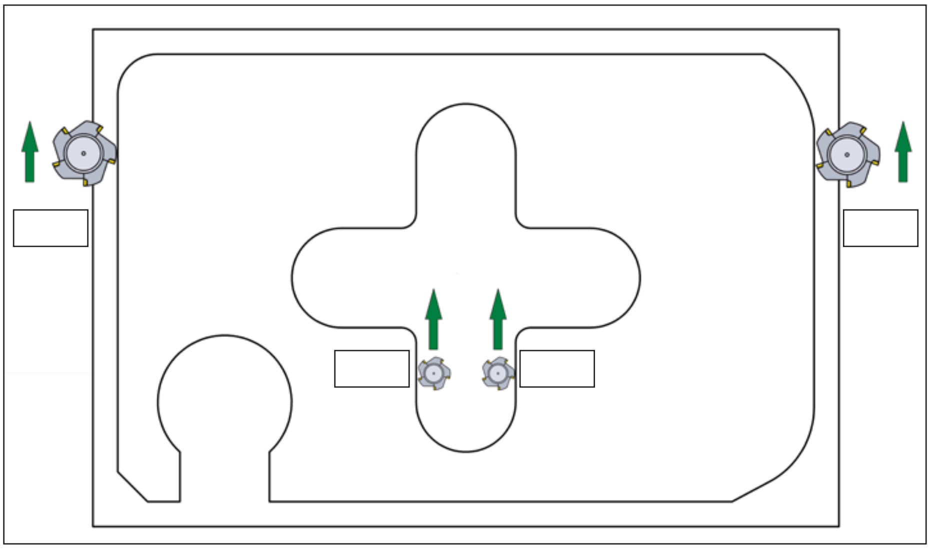
RL:

RR:

R0:



Add RL and RR to the figure.



Which machining direction is to be selected in order for contours to be machined with climb milling?

Inside contours:

Outside contours:

Which radius compensation is to be selected in order for contours to be machined with climb milling?

5.2 Approach and departure

APPR
DEP

5. Contour programming

Approach and departure

Explain the APPR and DEP functions.

APPR: __________

DEP: __________

Approach/departure point is a corner point

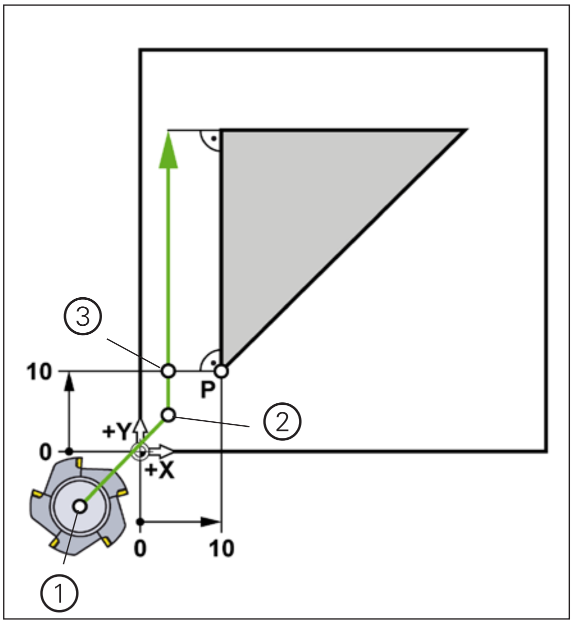
Program blocks 16 to 18 are being run.
Write down the coordinates of the starting point, of the calculated intermediate point and of the first calculated contour point.
The tool radius is 6 mm.

```
16 L X-7 Y-7 FMAX
17 APPR LT X+10 Y+10 LEN 5 RL
18 L Y+40
19 DEP LT LEN 18
20 L Z+100 FMAX
```

1: __________

2: __________

3: __________

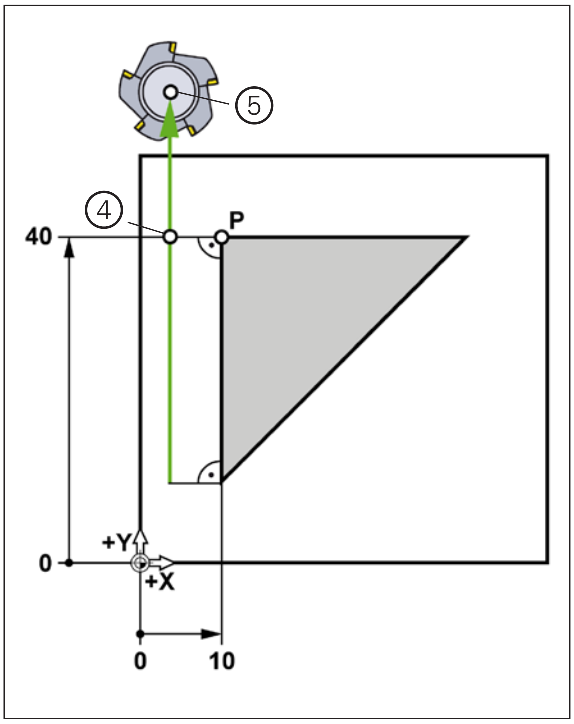


Program blocks 19 to 20 are being run.
Write down the coordinates of the last calculated contour point and of the calculated end point.

```
16 L X-7 Y-7
17 APPR LT X+10 Y+10 LEN 5 RL
18 L Y+40
19 DEP LT LEN 18
20 L Z+100 FMAX
```

4: ______________________

5: ______________________



DEP automatically rescinds the tool radius compensation.

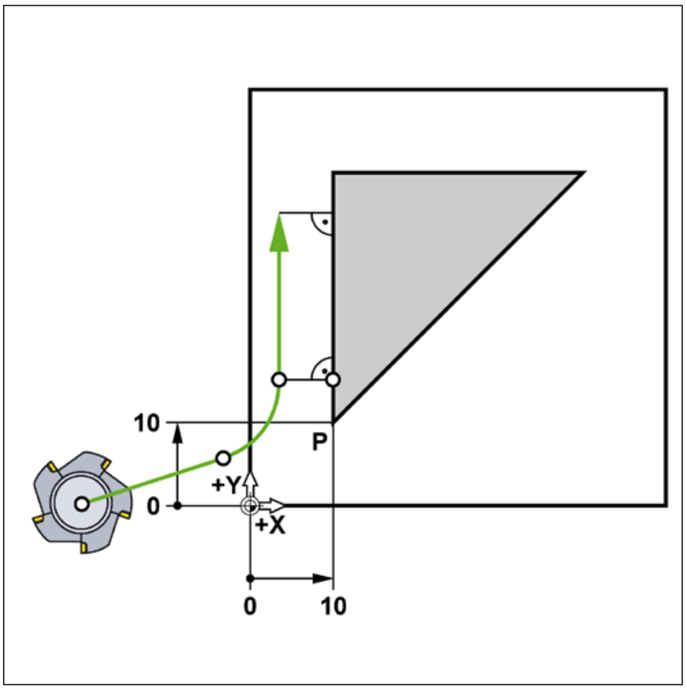
Optimize the Diamond.H program.

Approach/departure point lies on the contour

Complete program blocks 16 to 18.
The following conditions apply:

- Tool radius: 6 mm
- Pre-position to X-20, Y0
- Approach strategy: LCT
- 1st contour point: X+10, Y+15
- Approach radius: 10 mm
- 2nd contour point: X+10, Y+35

Use HEIDENHAIN Interactive Training as needed.



...	
16 L	Pre-positioning
17 APPR	Approach the first contour point with RL
18 L	Move to the second contour point
...	

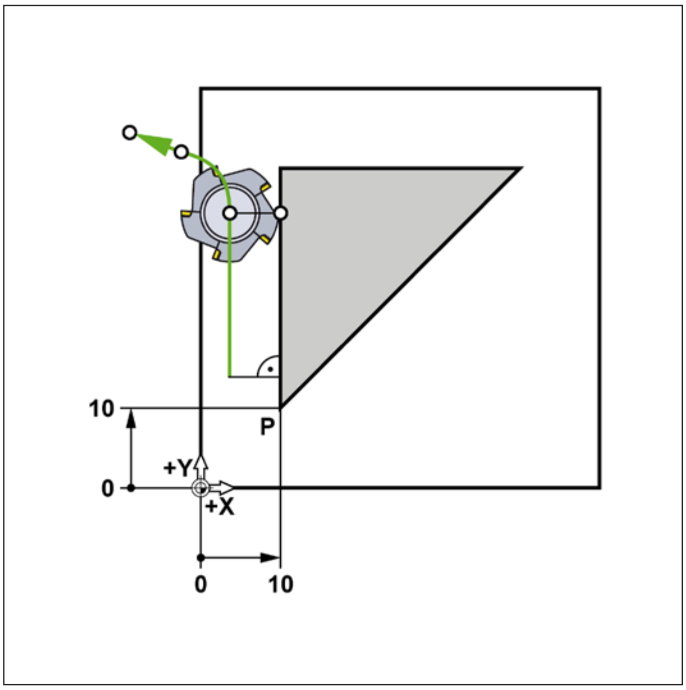
- If the approach/departure point lies on the contour, then mostly LCT (linear circular tangential) is used for approach/departure.

- If the approach/departure point is a corner point, then mostly LT (linear tangential) is used.

Complete program block 19.
The following conditions apply:

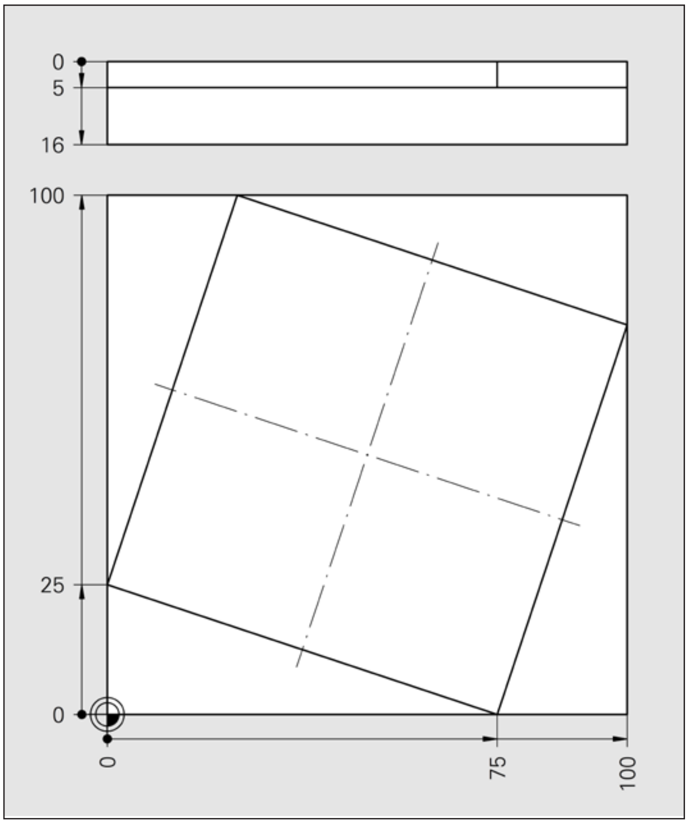
- Tool radius: 6 mm
- Depart to X-8, Y+45
- Departure strategy: LCT
- Departure radius: 8 mm

Use HEIDENHAIN Interactive Training as needed.



`...`	
`19 DEP`	Departure
`20 L Z+100 FMAX`	Move to clearance height
`...`	

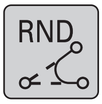
Write the program for the following workpiece on your programming station.
Starting point: X=75, Y=0
Use an appropriate approach and departure strategy.

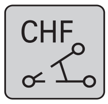


Inform yourself in TNCguide about the approach and departure strategies LN and CT.

HELP

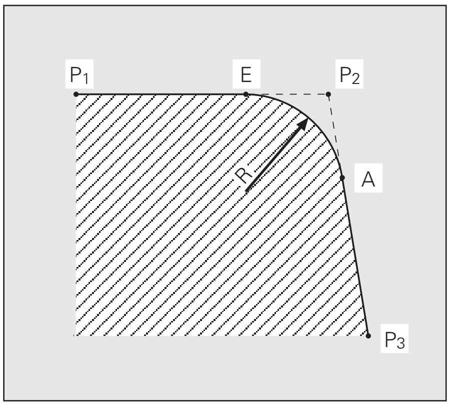
5.3 Roundings and chamfers

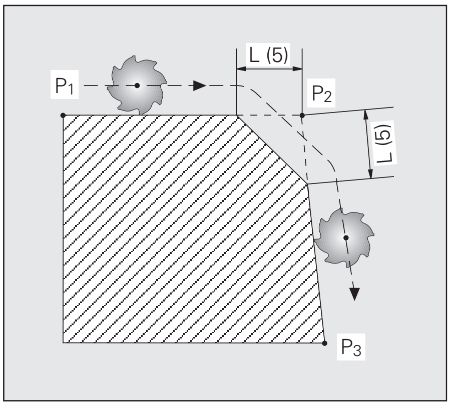




5. Contour programming

Roundings and chamfers





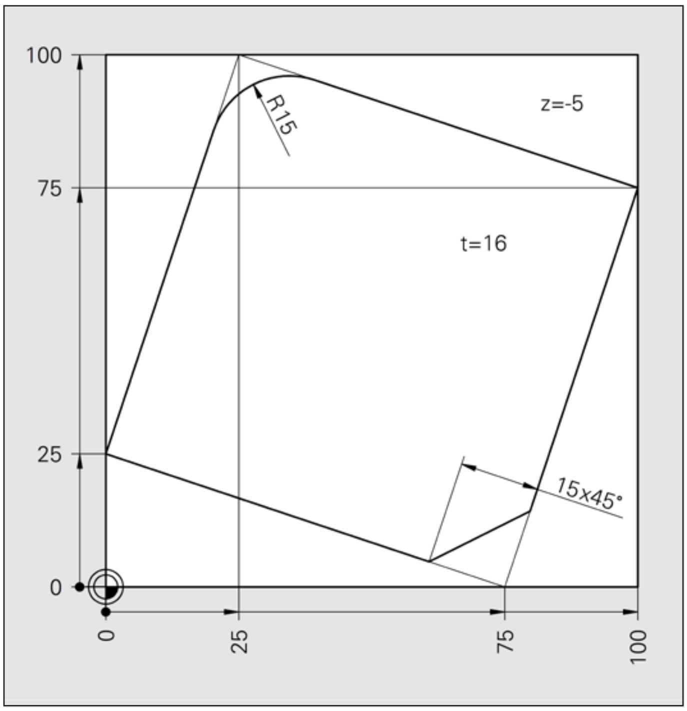
RND (=rounding) inserts a rounded connection between two straight lines or arcs.

`...`	
`10 L X... Y...`	Approach P1 as desired
`11 L X... Y...`	Approach P2
`12 RND R...`	Rounding arc
`13 L X... Y...`	Approach P3
`...`	

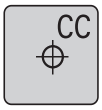
CHF (=chamfer) inserts a straight connection between two straight lines.

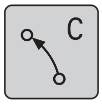
`...`	
`10 L X... Y...`	Approach P1 as desired
`11 L X... Y...`	Approach P2
`12 CHF 5`	Chamfer
`13 L X... Y...`	Approach P3
`...`	

Write the program for the following workpiece on your programming station.



5.4 Circles

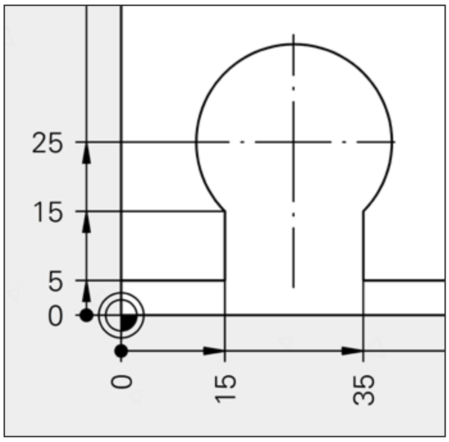




5. Contour programming

Circles CC/C

Circular motions can be programmed by entering the center point, end point and direction of rotation.
Explain the following entries.



CC: ______________________________

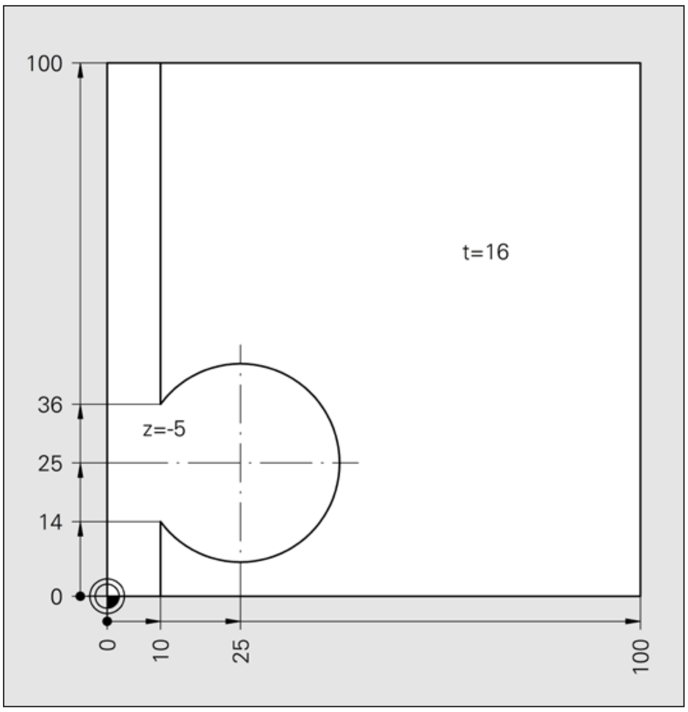
*DR+: ______________________________

*DR-: ______________________________

In the following example the circular arc of the drawing excerpt is milled with up-cut milling. Explain the three program blocks.

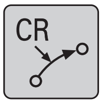
...	Explanation:
`10 L X+15 Y+15`	
`11 CC X+25 Y+25`	
`12 C X+35 Y+15 DR-`	
...	

Write the program for the following workpiece on your programming station.



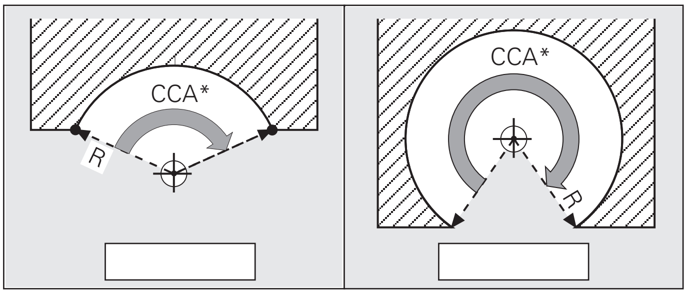
*DR=direction of rotation

5.5 Circles with radius



5. Contour programming

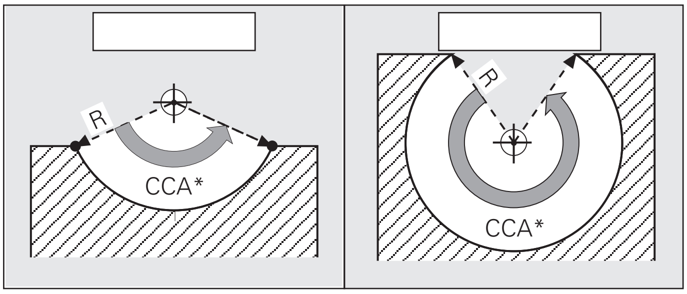
Circles with radius



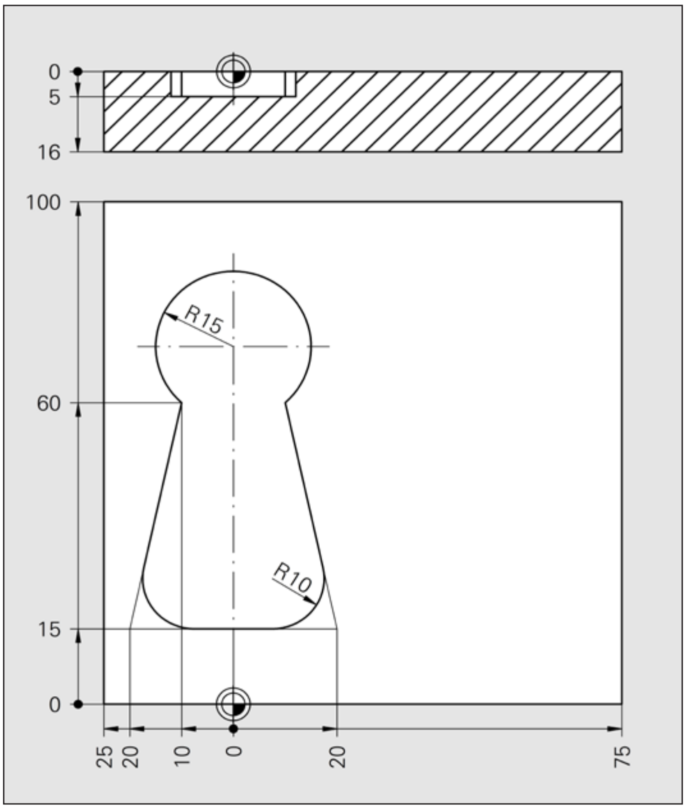
Remember that:

- Angle of CCA*<180°: sign for radius: R+
- Angle of CCA*>180°: sign for radius: R-
- Rotation clockwise: DR-
- Rotation counterclockwise: DR+

Add R+, R-, DR+, DR- to the four figures.

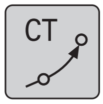


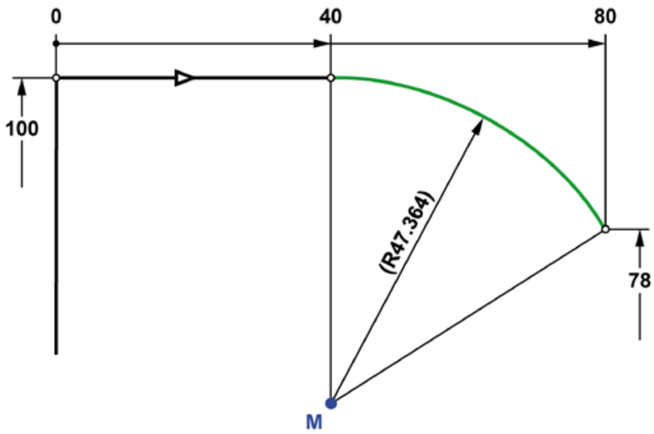
Write the program for the following contour on your programming station.



*CCA=circle center angle

5.6 Tangential circles





5. Contour programming

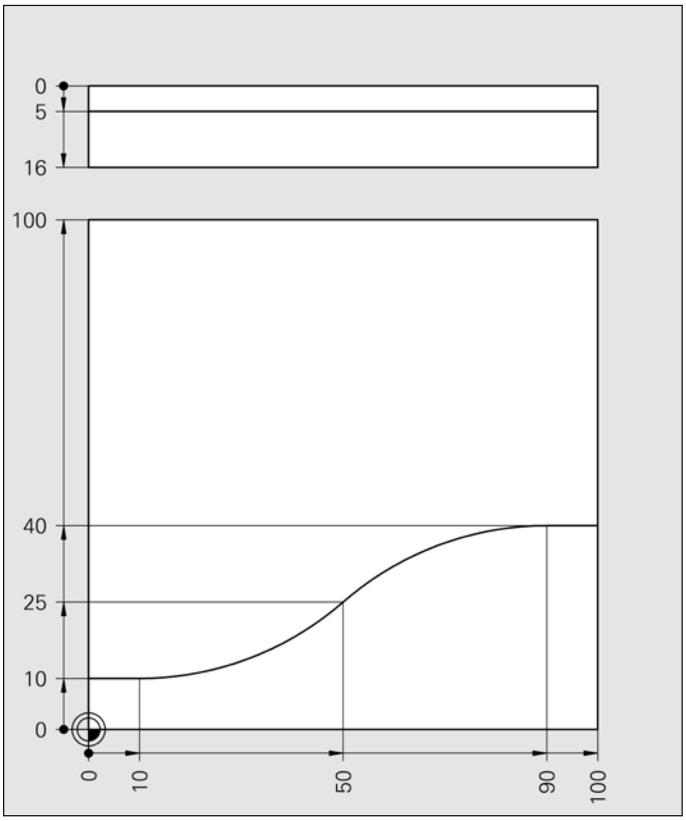
Tangential circles

What is meant by "tangential"?

Complete line 19.

`16 L X-30 Y+100`	Pre-position
`17 APPR LT X+0 Y+100 LEN 20 RL`	Approach
`18 L X+40`	Traverse motion to X+40
`19`	Circular path
`20 DEP LT LEN 10`	Departure

Write the program for the following contour on your programming station.



5. Contour programming

Arranging blocks

5. Contour programming

Summary

5. Contour programming

Writing NC programs

Enhance your knowledge with the Arranging blocks, Summary and Writing NC programs sections. These sections prepare you for the second test.

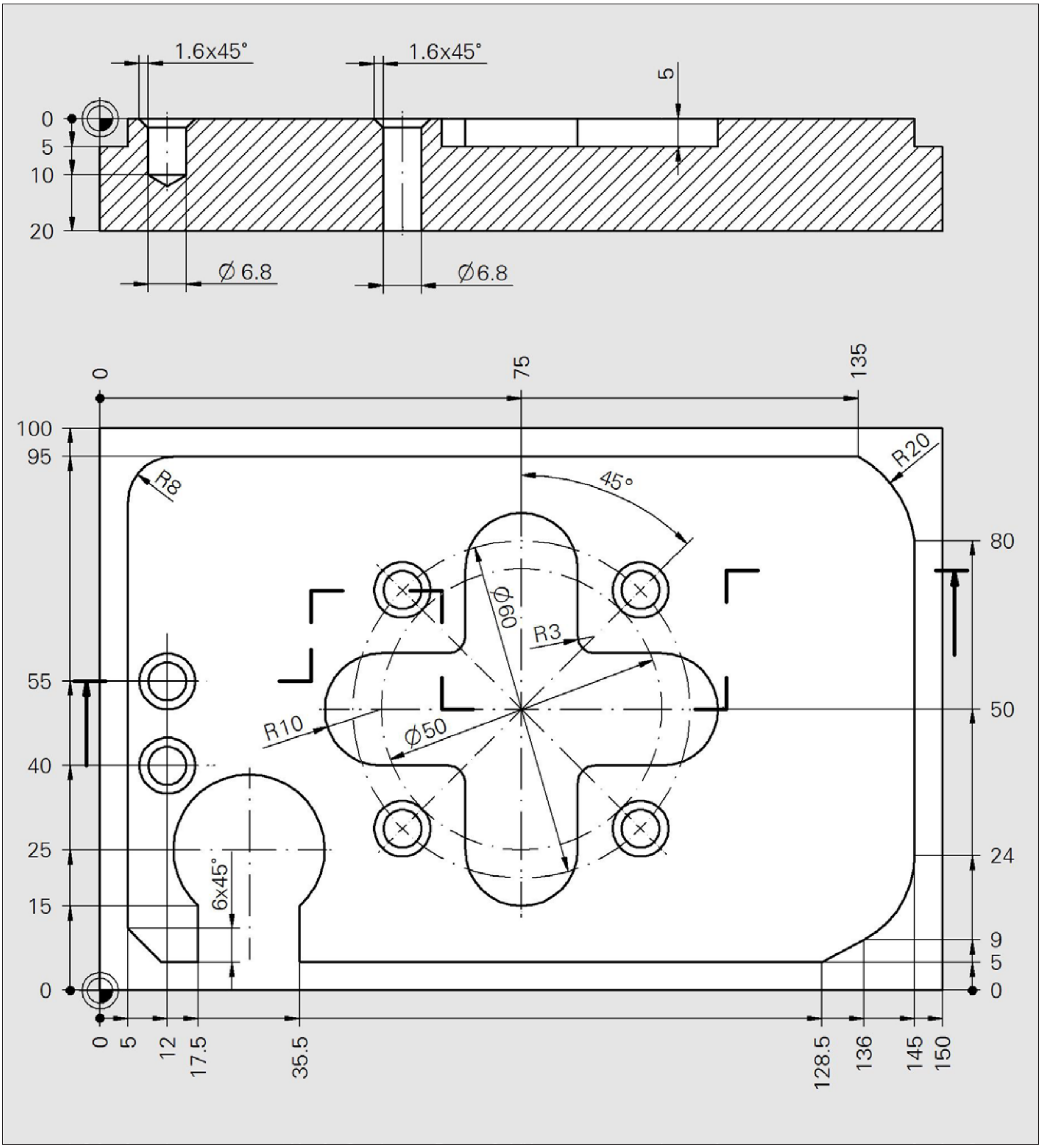
5. Contour programming

Test 2

Complete the second test and print it out. Paste it in here for your documentation. Paste it over the informational text on this page. You can repeat the test as many times as you want until you are satisfied with the result.

Programming station

Start the programming station.
Add the inside and outside contours to your program for the "pattern plate" workpiece.



Exercises 5a, b, c, d

6

FK programming

6 FK programming

6. FK programming

Basic functions, Application

Programming a contour

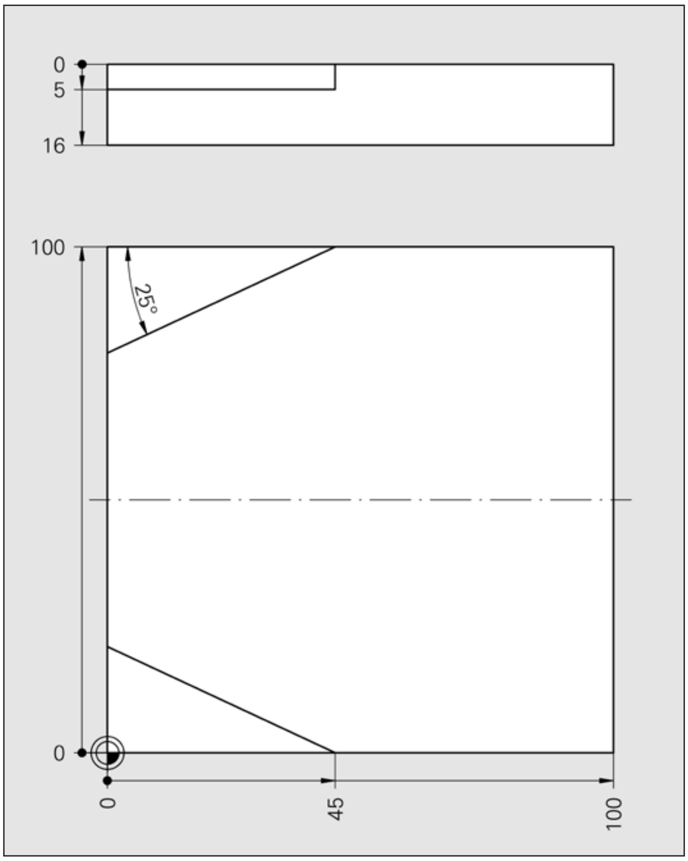
FK = free contour

If the workpiece drawing is not dimensioned for NC, and therefore cannot be programmed with the gray path-function keys, FK programming comes to your aid.

In the "angled workpiece" example, only the X coordinate and rise angle are known for the lower diagonal.
The Y coordinate is missing.
In order to program this contour using the functions described until now, you would have to calculate the Y value using trigonometric functions.

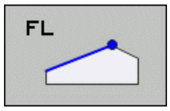
This is not necessary with FK.

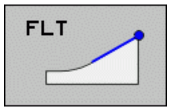
There are four basic functions.
Explain them.



FL: ______________________________

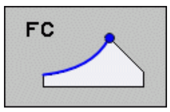
FLT: ______________________________

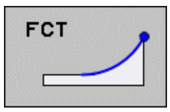




FC: ______________________________

FCT: ______________________________





TNCguide has more detailed information about FK.

Complete blocks 8 to 11 of the “angled workpiece” program.

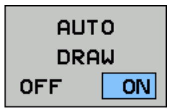
0 BEGIN PGM ANGLED_WORKPIECE MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	
4 L Z+100 R0 FMAX M3	
5 L X+110 Y-10 R0 FMAX	
6 L Z-5 R0 FMAX	
7 APPR LT X+100 Y+0 LEN10 RL F1000	
8 L X	
9 FL X AN	* X value and rise angle known
10 FL AN	* Rise angle known
11 FL X Y AN	* X value, Y value and rise angle known
12 L X+100	
13 L Y+0	
14 DEP LT LEN10	
15 L Z+100 R0 FMAX M30	
16 END PGM ANGLED_WORKPIECE MM	

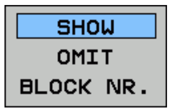
Program the workpiece on the programming station using the FK functions.

Note the following settings:

In the **Programming** operating mode, switch to the PROGRAM+GRAPHICS view using the screen layout key.

In the third soft-key row, toggle the two soft keys at right to ON and SHOW.



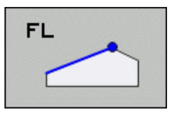


* Initiate FK dialogs with the FK key.

Enter all available data for every contour element.
Example for line 9:

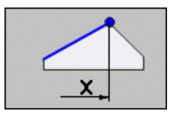
▶ Preliminary considerations:

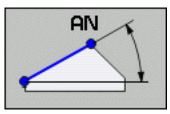
- What does the contour element look like? Straight line
- How does the the contour element start? Not tangential
- Press the appropriate soft key. FL



▶ Enter all known values via soft keys.

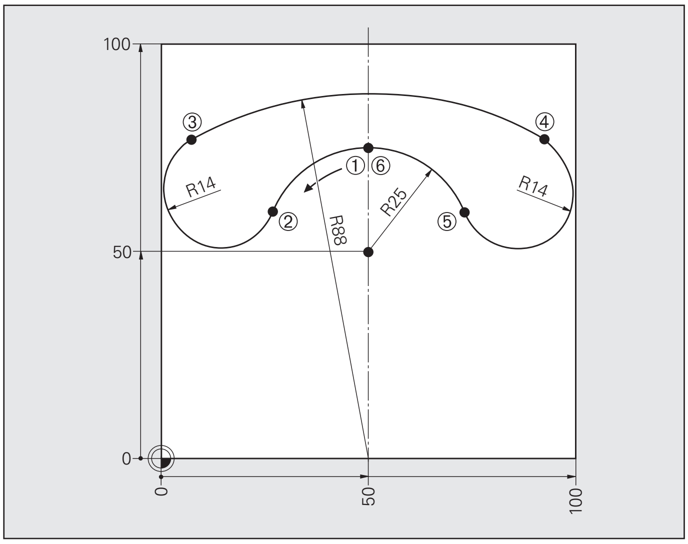
- X value known: enter value via soft key
- AN known: enter value via soft key





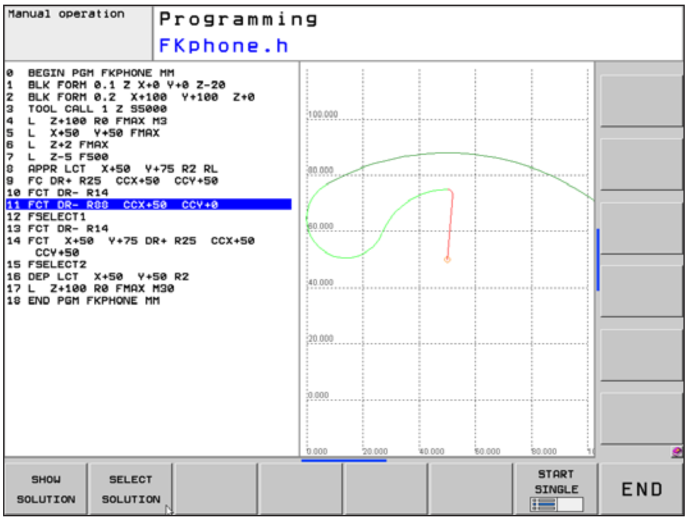
Assignment: Telephone
Complete program blocks 9, 10, 11, 13 and 14!
For line 11, please refer to the next page.
Then program the contour on the programming station!
(workpiece blank dimensions: 100x100x20, contour as island, Z-5)

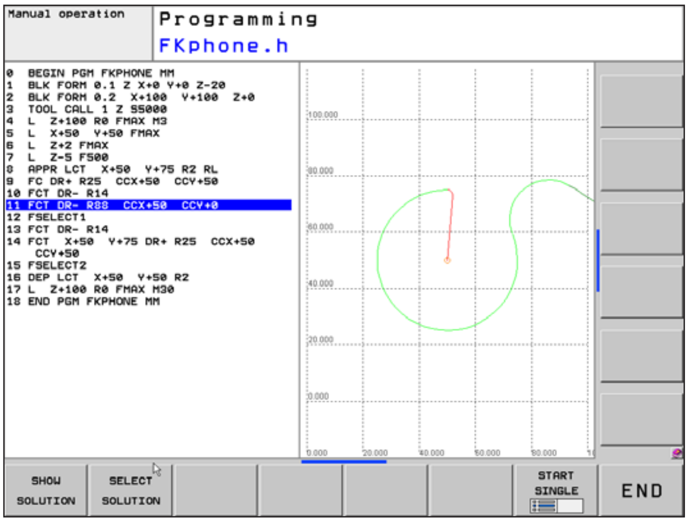
Which basic FK functions are needed in this example?



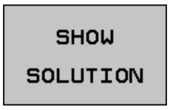
0 BEGIN PGM TELEPHONE MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-10	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	
4 L Z+100 R0 F MAX M3	
5 L X+50 Y+50 R0 F MAX	
6 L Z+2 F MAX	
7 L Z-5 F AUTO	
8 APPR LCT X+50 Y+75 R2 RL F500	Point 1
9 FC DR R CCX CCY	Point 2
10 FCT DR R	Point 3
11 FCT DR R CCX CCY	Point 4
12 FSELECT 1	
13 FCT DR R	Point 5
14 FCT X Y DR R CCX CCY	Point 6
15 FSELECT 2	
16 DEP LCT X+50 Y+50 R2	
17 L Z+100 R0 F MAX M2	
18 END PGM TELEPHONE MM	

Mathematically there are two possibilities for line 11, but only one is correct:

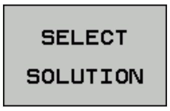




Press the SHOW SOLUTION soft key to cycle through the possibilities.



Select the correct contour element as shown in the drawing.
The TNC automatically inserts block 12.

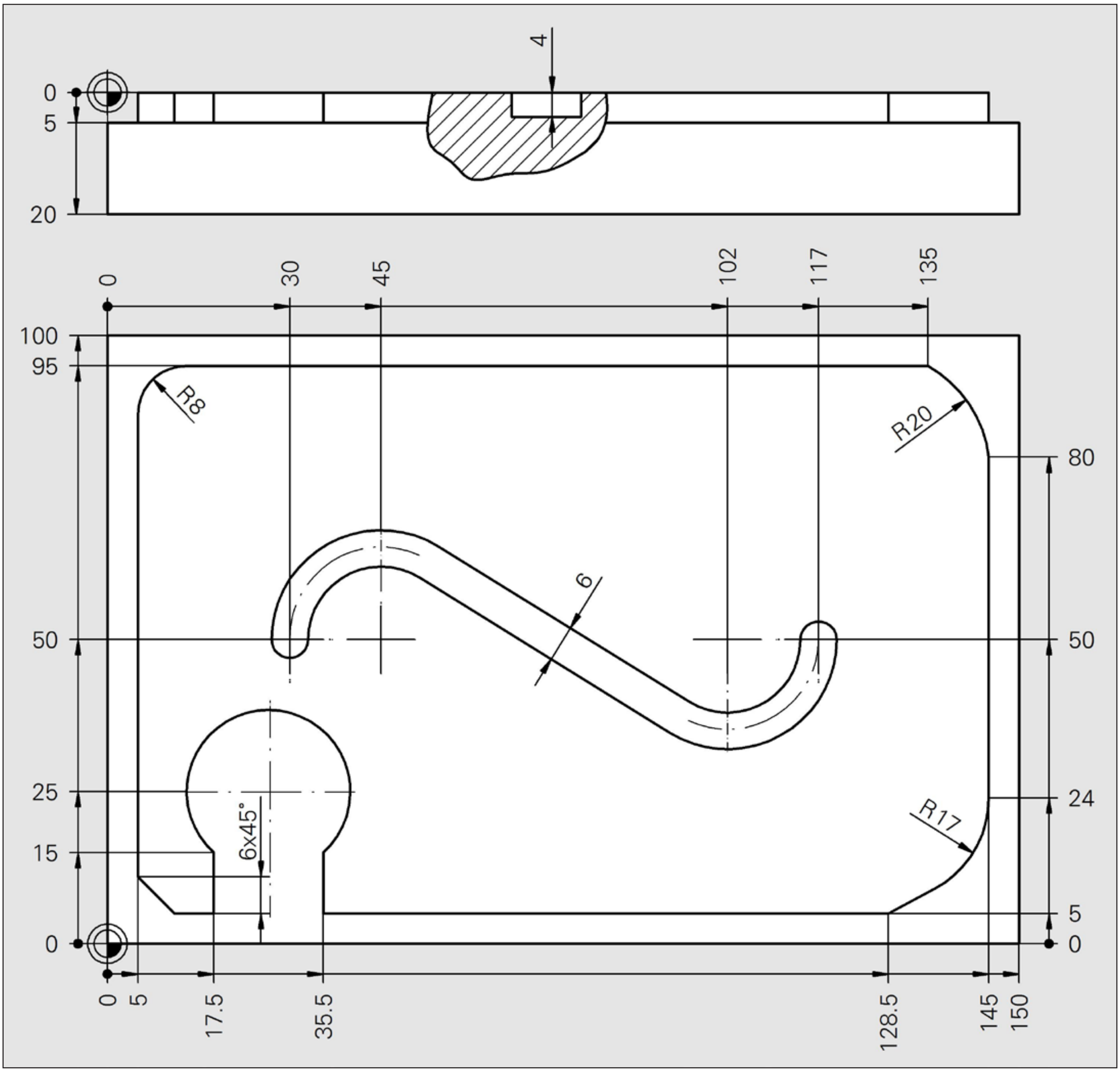


Meaning of the color depiction of the contour elements (depends on the machine):

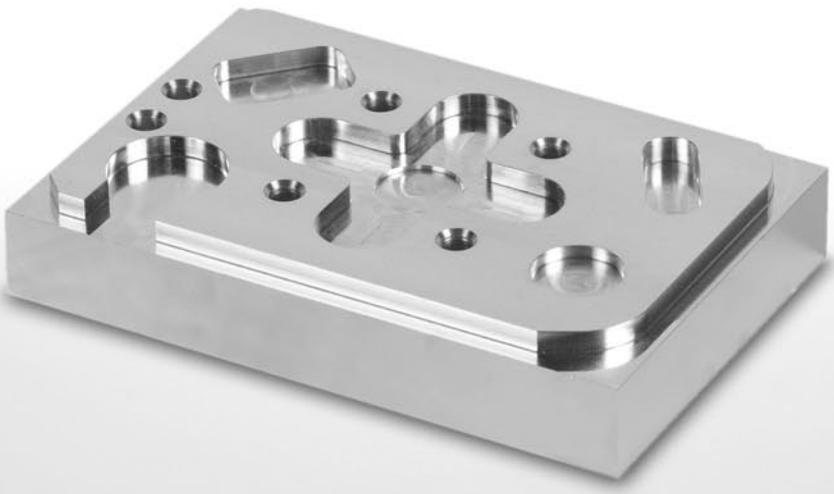
Color	Function
Black	Contour element is fully defined.
Blue	More than one solution is possible for the entered data.
Red	More data is required to calculate the contour or contour element.
Green	More than one solution is possible. These are shown via the SHOW SOLUTION soft key.

Programming station

Start the programming station.
Create a new program and write the program for this workpiece.



Exercises 6a, b, c, d



7

Cycles

7 Cycles

7. Cycles

CYCL DEF / CYCL CALL

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles.
This makes it very simple to program slots, rectangular pockets, drill patterns, etc.

Explain CYCLE DEF and CYCLE CALL.

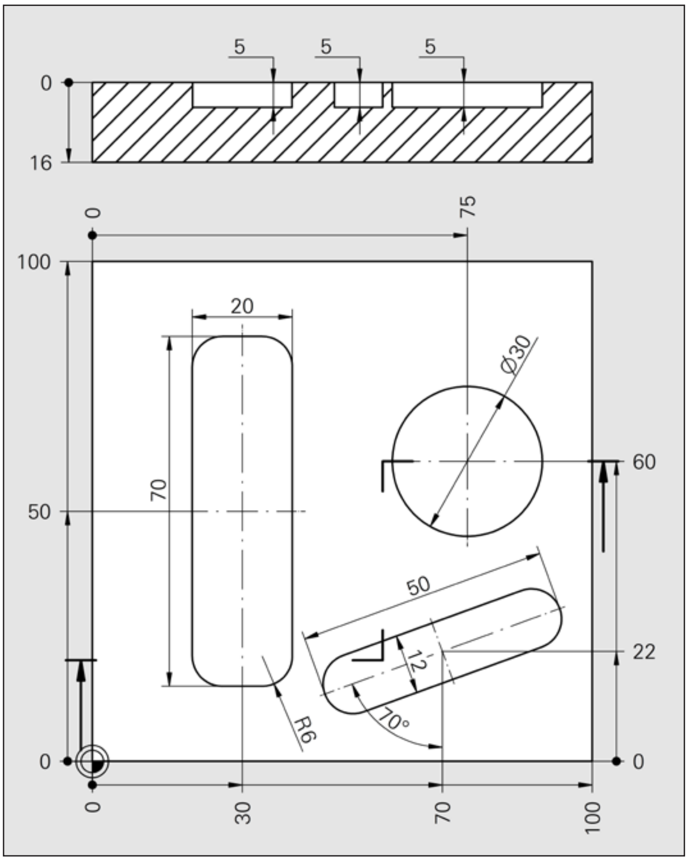
CYCLE DEF: ______________________________

CYCLE CALL: ______________________________

7. Cycles

Face milling, Rectangular pocket, Circular pocket, Slot milling

Program the following workpiece on the programming station based on the scheme on the next page.

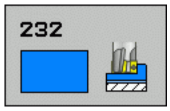


TNCguide contains comprehensive information.
All parameters of the cycles are explained there.

HELP

Inputting line 5:

- Press the CYCLE DEF key.
- Press the MULTIPASS MILLING soft key.
- Select Cycle 232.

CYCL DEF ①   MULTIPASS MILLING ②    ③

Inputting line 6:

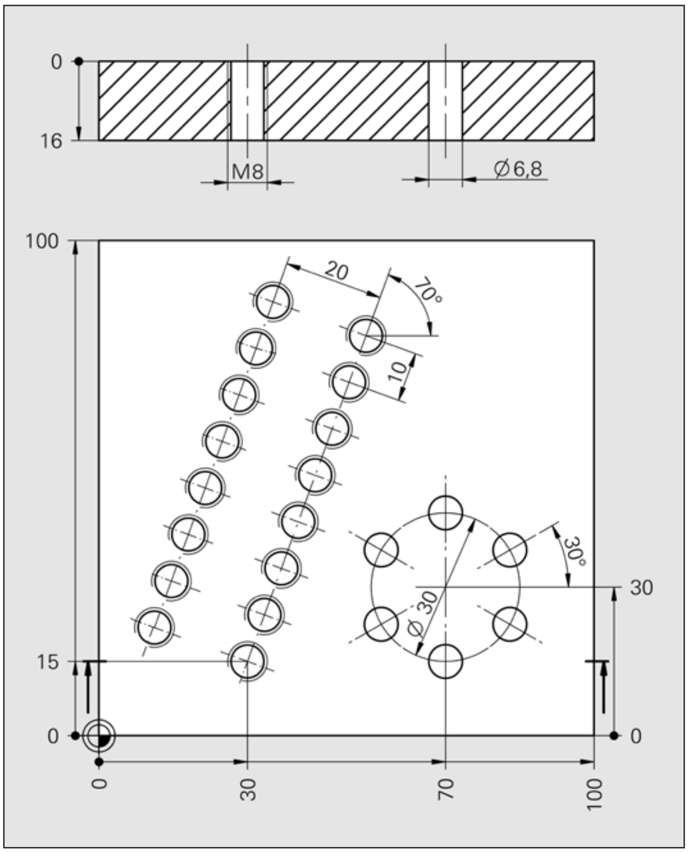
- Press the CYCLE CALL key.
- Press the CYCLE CALL M soft key.

CYCL CALL ①   CYCLE CALL M ②

`0 BEGIN PGM PLAN MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-16`	
`2 BLK FORM 0.2 X+100 Y+100 Z+2`	Unmachined upper surface: Z=2
`3 TOOL CALL 20 Z S2000 F300`	
`4 L Z+100 R0 FMAX M3`	
`5 CYCL DEF 232 FACE MILLING ~`	Face milling definition
`Q389=+2 ;STRATEGY ~`	
`Q225=+0 ;STARTNG PNT 1ST AXIS ~`	
`Q226=+0 ;STARTNG PNT 2ND AXIS ~`	
`Q227=+2 ;STARTNG PNT 3RD AXIS ~`	
`Q386=+0 ;END POINT 3RD AXIS ~`	After face milling: upper surface Z=0
`...`	
`...`	
`6 CYCL CALL`	Call face milling
`7 L Z+100 R0 FMAX`	
`8 TOOL CALL 5 Z S5000 F300`	
`9 L Z+100 R0 FMAX M3`	
`10 L X+30 Y+50 R0 FMAX`	Positioning for rectangular pocket
`11 CYCL DEF 251 RECTANGULAR POCKET ~`	Definition of rectangular pocket
`...`	
`...`	
`12 ...`	Call rectangular pocket
`13 L Z+100 R0 FMAX`	
`14 L X... Y... R0 FMAX`	Positioning for circular pocket
`15 CYCL DEF 252 CIRCULAR POCKET ~`	Definition of circular pocket
`...`	
`...`	
`16 ...`	Call circular pocket
`17 L Z+100 R0 FMAX`	
`18 L X ...Y ... R0 FMAX`	Positioning for slot
`19 CYCL DEF 253 SLOT MILLING ~`	Definition of slot
`...`	
`...`	
`20 ...`	Call slot
`21 L Z+100 R0 FMAX M30`	
`22 END PGM PLAN MM`	

Drilling, Rigid tapping, Universal drilling

Program the following workpiece on the programming station based on the scheme on the next page.

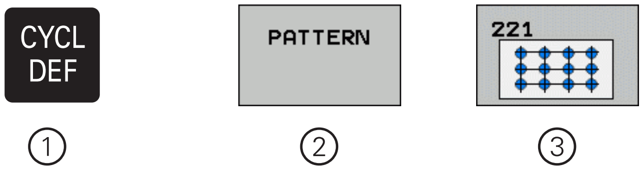


Cycles 221 (Cartesian Pattern) and 220 (Polar Pattern) each call the last defined cycle at the defined positions.

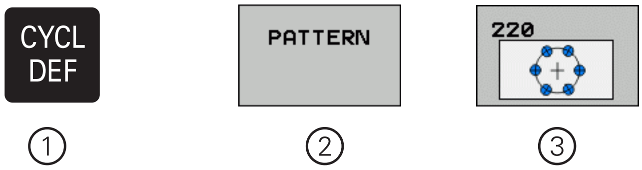
All cycles which remove material (such as drilling) must be called (CALL-active cycles).

Cycles which do not remove material (such as coordinate transformation cycles) do not need to be called (DEF-active cycles).

Inputting line 6:



Inputting line 16:

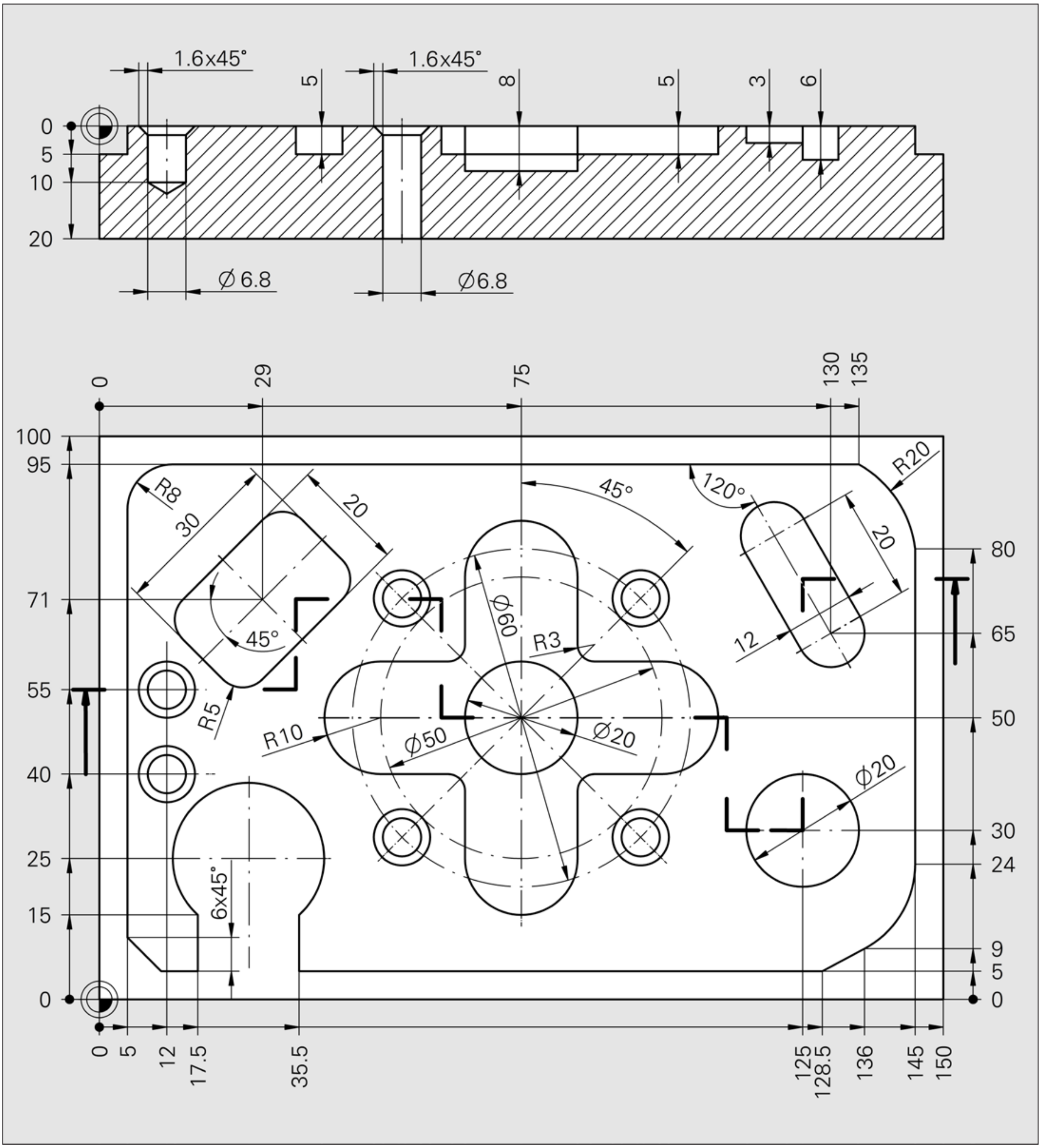


0 BEGIN PGM THREAD MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-16	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S1111 F555	
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 200 DRILLING ~	Definition of drill holes
...	
...	
6 CYCL DEF 221 CARTESIAN PATTERN ~	Call of drill holes on Cartesian pattern
...	
...	
7 L Z+100 R0 FMAX	
8 TOOL CALL 2 Z S1111 F555	
9 L Z+100 R0 FMAX	
10 CYCL DEF 207 RIGID TAPPING NEW ~	Definition of thread
...	
...	
11 CYCL DEF 221 CARTESIAN PATTERN ~	Call of thread on Cartesian pattern
...	
...	
12 L Z+100 R0 FMAX	
13 TOOL CALL 1 Z S1111 F555	
14 L Z+100 R0 FMAX M3	
15 CYCL DEF 203 UNIVERSAL DRILLING ~	Definition of drill holes on polar pattern
...	
...	
16 CYCL DEF 220 POLAR PATTERN ~	Call of drill holes on polar pattern
...	
...	
17 L Z+100 R0 FMAX M30	
18 END PGM THREAD MM	

TNCguide contains comprehensive assistance.
All cycles as well as the possibilities for calling cycles are explained there.

Programming station

Start the programming station.
Add the cycles to your program for the “pattern plate” workpiece.



Exercises 7a, b, c, d

8

LBL applications

8 LBL applications

8.1 LBL as program section repeat

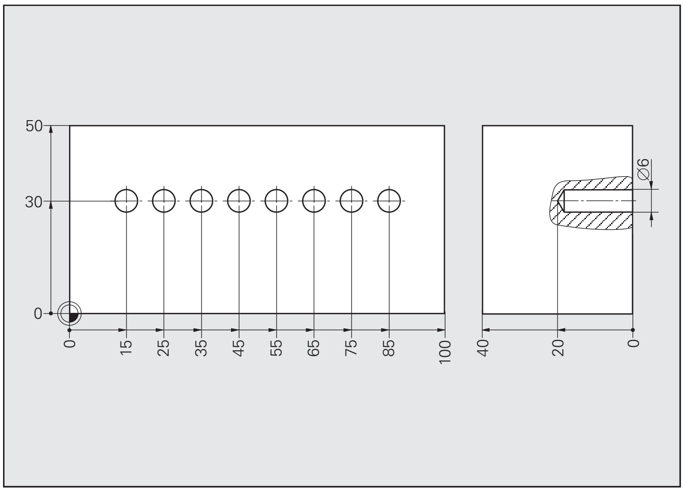
Programming loops

8. LBL Applications

Program-section repeat

Program section repeats are well suited for identical elements (such as drill holes) that are equally spaced.

A LABEL is set to mark a block number, starting from which a section of the program is repeated.



This is done with the LBL SET key. Labels receive either a number (1 to 999) or a name.
The end of the program section to be repeated is defined with the LBL CALL key. The number of repetitions (REP) is entered here.
Write the “LINEAR_PATTERN” program on the programming station.

Six returns (REP6) to line 8

`0 BEGIN PGM LINEAR_PATTERN MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	
`2 BLK FORM 0.2 X+100 Y+50 Z+0`	
`3 TOOL CALL 3 Z S3000 F300`	
`4 L Z+100 R0 FMAX M3`	
`5 CYCL DEF 203 UNIVERSAL DRILLING ~`	Definition of drilling cycle
`...`	
`6 L X... Y... R0 FMAX`	Approach position 1 (X=15, Y=30)
`7 CYCL CALL`	Call drilling cycle at position 1
`8 LBL 1`	**Set label**
`9 L IX... R0 FMAX`	Incremental motion to next position
`10 CYCL CALL`	Call drilling cycle at positions 2 to 8
`11 CALL LBL 1 REP6`	**Label call with six repeats**
`12 L Z+100 R0 FMAX M30`	
`17 END PGM LINEAR_PATTERN MM`	

Simulate the program in the **Test Run** operating mode one block at a time.
You can see each of the jumps.

8.2 LBL as subprogram

Repetition of identical elements

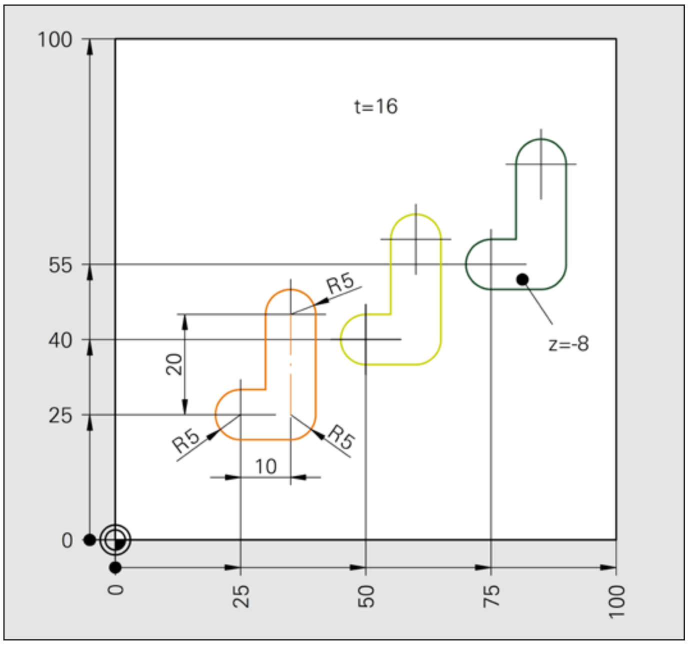
Elements that are repeated (contours, actions called on pitch circles, ...) can easily be programmed using subprograms.

Subprograms are entered after the end of the main program (after M30 or M2).
Subprograms are initiated with the LBL SET key, and are given a number (e.g., 1).

LBL SET

Subprograms are called with the LBL CALL key.
The NO ENT key is pressed in response to the REP question.

LBL CALL



LBL 0 (line 23) means:
- End of subprogram
- Return to main program

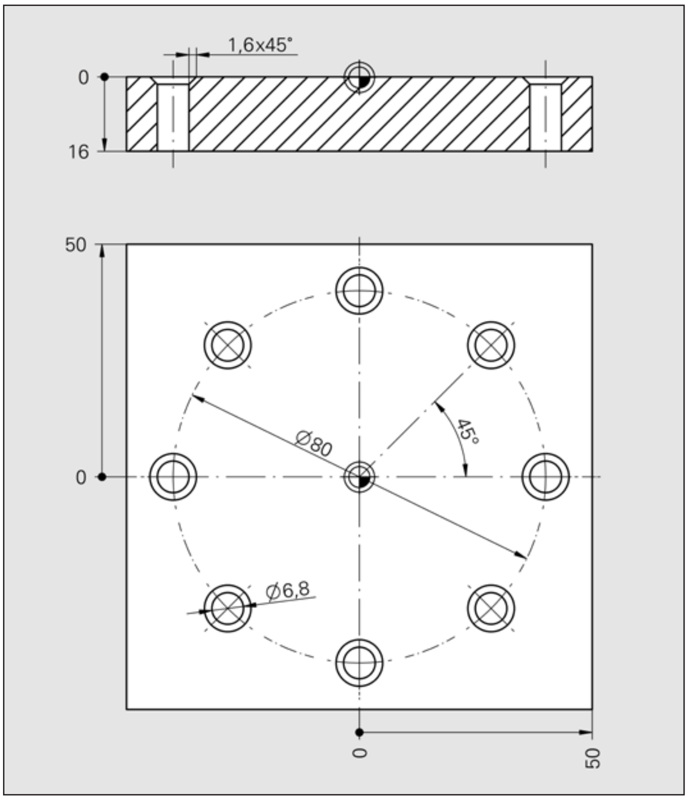
Main program

```
0  BEGIN PGM SP MM
1  BLK FORM 0.1 Z  X+0  Y+0  Z-16
2  BLK FORM 0.2  X+100  Y+100  Z+0
3  TOOL CALL 5 Z S2000 F250
4  L  Z+100 R0 FMAX M3
5  L  Z+2 R0 FMAX
6  * -
7  L  X+25  Y+25 R0 FMAX
8  CALL LBL 1
9  * -
10 L  X+50  Y+40 R0 FMAX
11 CALL LBL 1
12 * -
13 L  X+75  Y+55 R0 FMAX
14 CALL LBL 1
15 * -
16 L  Z+100 R0 FMAX M30
17 * -
18 LBL 1
19 L Z-8 R0 F AUTO
20 L IX+10
21 L IY+20
22 L Z+2
23 LBL 0
24 END PGM SP MM
```

Subprogram

Cycle call on pitch circle

A pitch circle is programmed in this example.
The cycles (counterboring, drilling) are called in a subprogram.

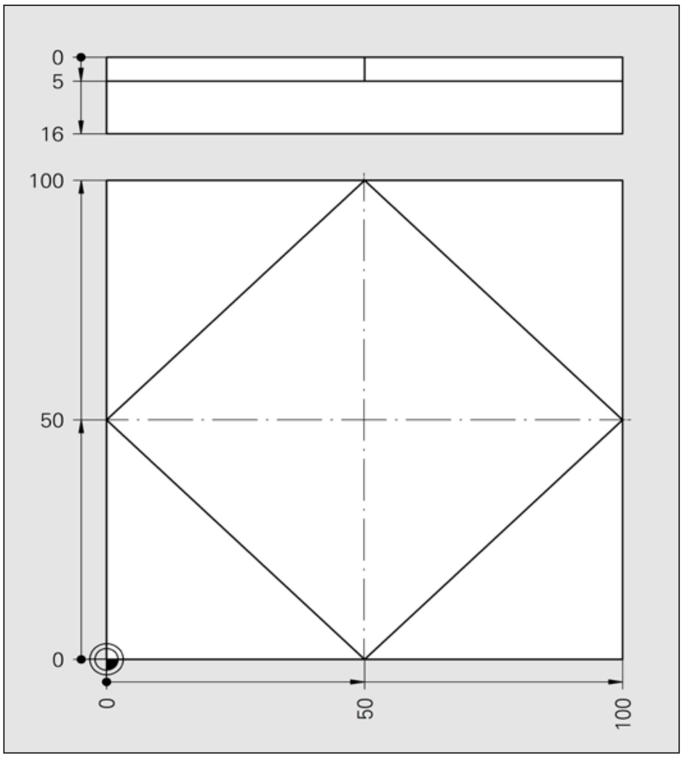


Write the program on the programming station.

0 BEGIN PGM SP2 MM	
1 BLK FORM 0.1 Z X-50 Y-50 Z-16	
2 BLK FORM 0.2 X+50 Y+50 Z+0	
3 TOOL CALL 1 Z S2000 F200	
4 L Z+100 R0 FMAX M3	
5 CYCL DEF 200 DRILLING ~	Definition of counterbores
...	
6 CALL LBL 1	Call LBL1, counterboring
7 L Z+100 R0 FMAX	
8 TOOL CALL 2 Z S2620 F511	
9 L Z+100 R0 FMAX M3	
10 CYCL DEF 203 UNIVERSAL DRILLING ~	Definition of drill holes
...	
11 CALL LBL 1	Call LBL1, drilling
12 L Z+100 R0 FMAX M30	
13 * -	
14 LBL 1	Beginning of subprogram
15 CYCL DEF 220 POLAR PATTERN ~	
...	
16 LBL 0	End of subprogram, return to line 7 / 12
17 END PGM SP2 MM	

Roughing and finishing

Another important application is the roughing and finishing of a contour. The contour description as well as the approach/departure instructions are written in a subprogram.



Write the program on the programming station.

0 BEGIN PGM ROUGHFINISH2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-16	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 10 Z S5000 F500 DL+0.3 DR+0.3	Call roughing cutter with oversize
4 L Z+100 R0 FMAX M3	
5 L X-15 Y+35 R0 FMAX	
6 L Z+2 R0 FMAX	
7 L Z-5 R0 F AUTO	
8 CALL LBL 1	Call subprogram
9 * -	
10 TOOL CALL 8 Z S5000 F400	Call finishing cutter
...	
15 CALL LBL 1	Call subprogram
16 * -	
17 L Z+100 R0 FMAX M30	End of main program
18 * -	
19 LBL 1	Subprogram with contour description
20 APPR LT X+0 Y+50 LEN10 RL	
...	
25 DEP LT LEN10	
26 LBL 0	End of subprogram, return to main program
27 END PGM ROUGHFINISH2 MM	

Adding a chamfer to a contour

The addition of a chamfer is programmed using the same strategy as for roughing and finishing.
A 1 mm chamfer is to be added to the border of the diamond on the previous page.

Remember the following:

- Enter the chamfer cutter (90°) in the tool table, with a radius of 2 mm, for example.
- A milling depth of 3 mm results in a chamfer width of 1 mm.

What chamfer width results from a radius of 3 mm and a milling depth of 3.5 mm?

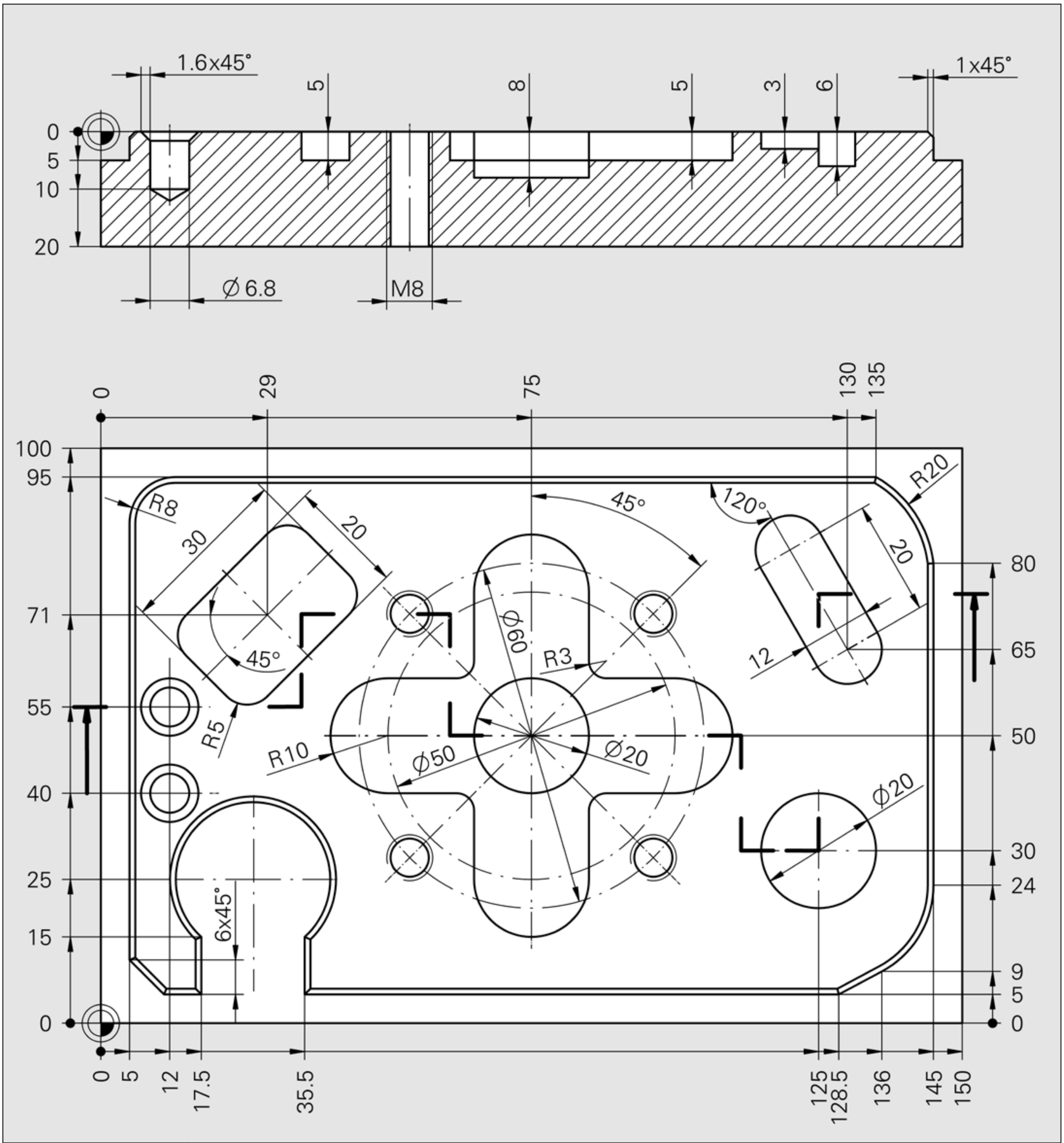
Write the program on the programming station.

`0 BEGIN PGM CONTOUR_CHAMFER MM`	
`...`	
`3 TOOL CALL 12 Z S2500 F250`	Call end mill
`...`	
`7 L Z-5 R0 F AUTO`	
`8 CALL LBL 1`	Call subprogram
`9 L Z+100 R0 FMAX`	
`10 TOOL CALL 1 Z S2500 F250`	Call chamfer cutter (radius: 2 mm)
`...`	
`14 L Z-3`	Milling depth: 3 mm, chamfer: 1 mm
`15 CALL LBL 1`	Call subprogram
`16 L Z+100 R0 FMAX M30`	
`17 * -`	
`18 LBL 1`	Subprogram with contour description
`19 APPR LT X+0 Y+50 LEN10 RL F AUTO`	
`...`	
`24 DEP LT LEN10`	
`25 LBL 0`	End of subprogram, return to main program
`26 END PGM CONTOUR_CHAMFER MM`	

Chamfers cannot be displayed in the **`Test Run`**.

Optimize the "pattern plate" program as follows.

- Face milling of the workpiece blank
- Pitch circle with subprogram
- Roughing and finishing of the outside contour, adding a chamfer, in subprogram



Exercises 8a, b, c, d

Final test

Final test

Take the final test in the HEIDENHAIN Interactive Training program and print it out.

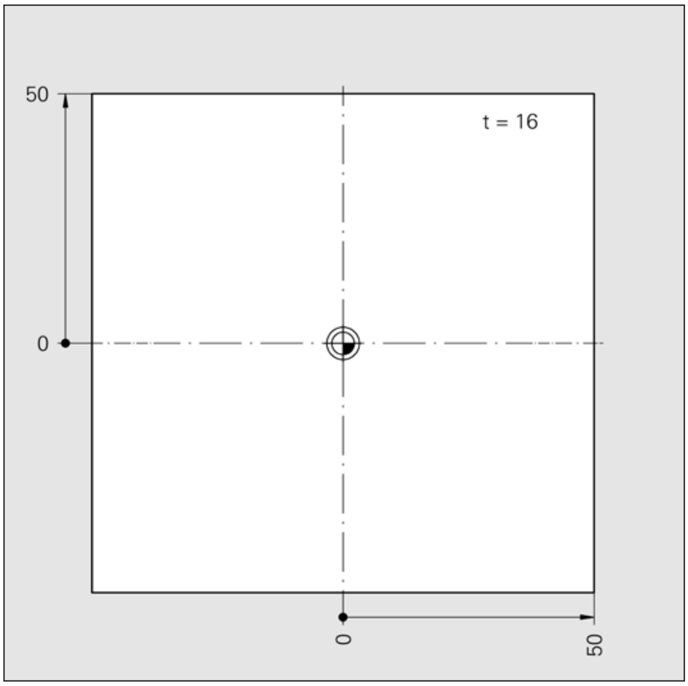
Paste it in here for your documentation.
Paste it over the informational text on this page.
You can repeat the test as many times as you want until you are satisfied with the result.

9

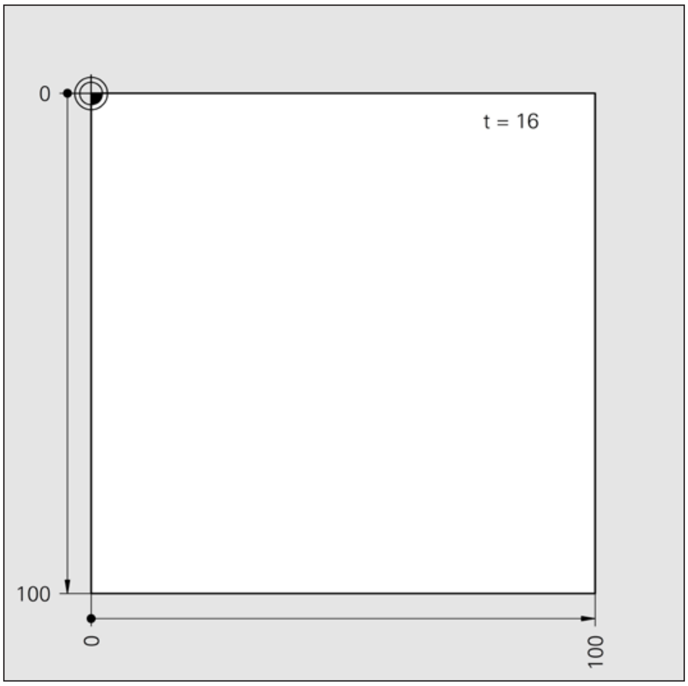
Appendix

9 Appendix

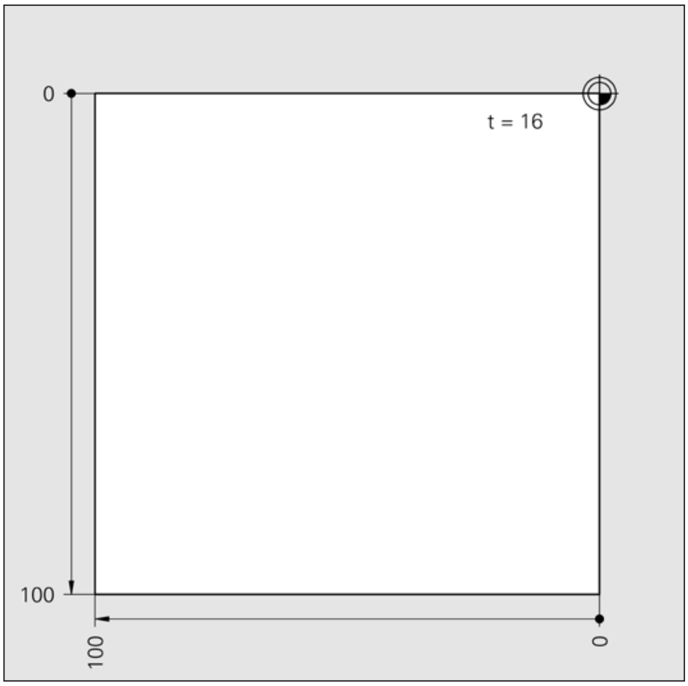
Exercise 2a



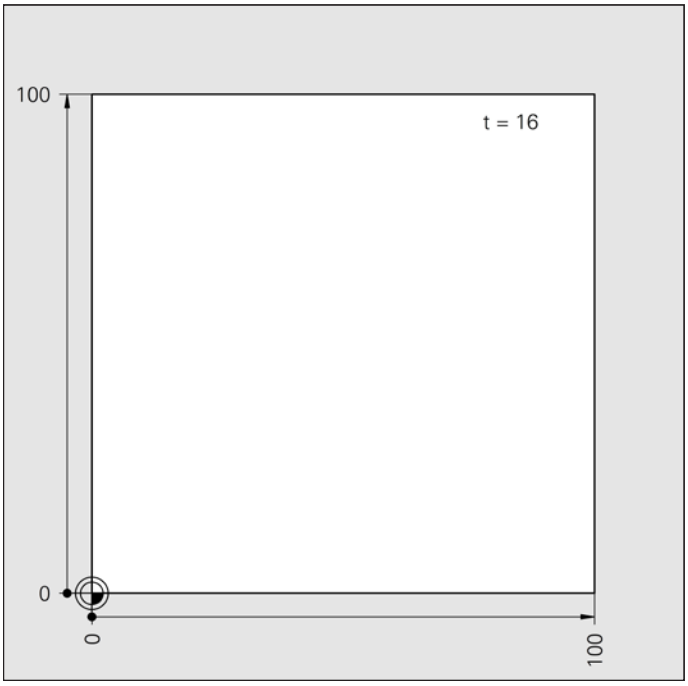
Exercise 2b



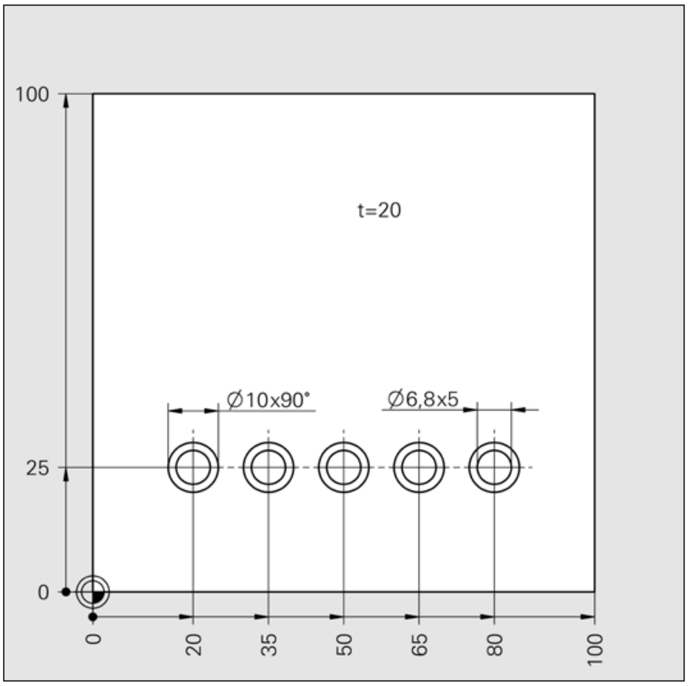
Exercise 2c



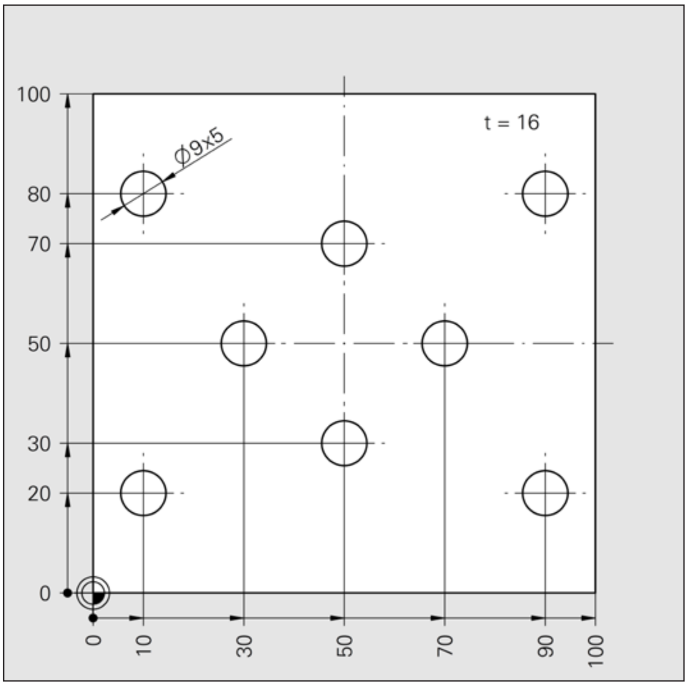
Exercise 2d



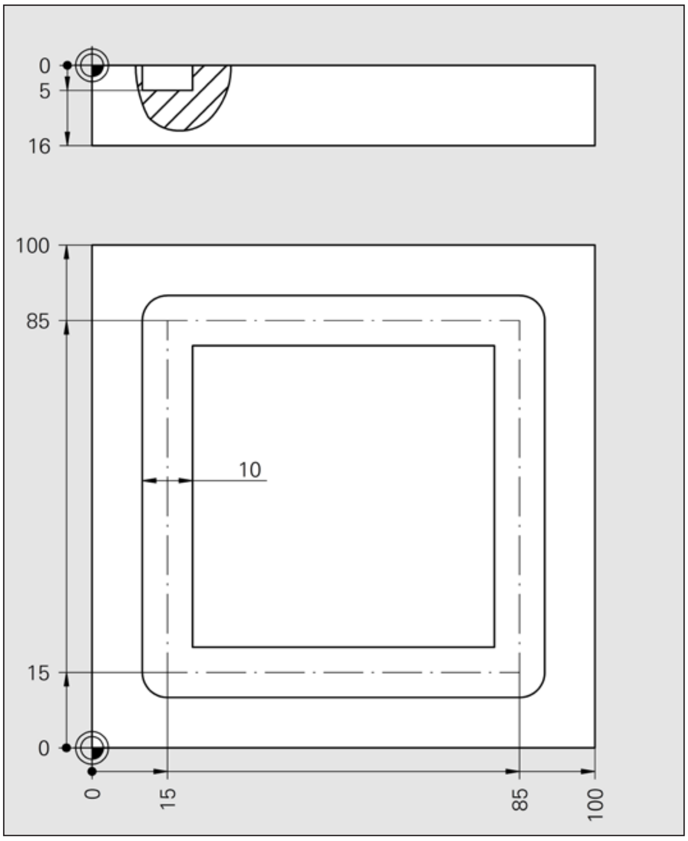
Exercise 4a



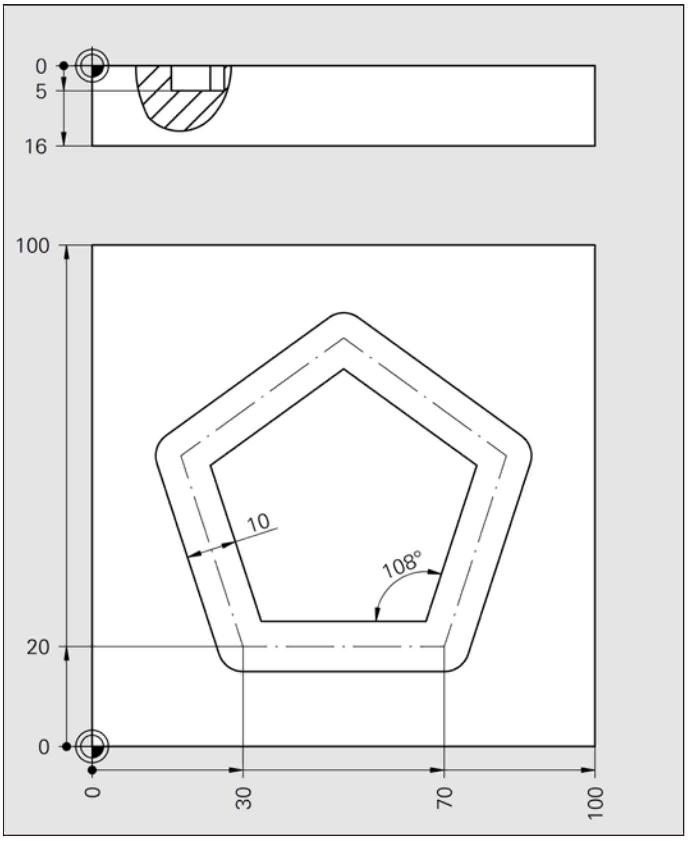
Exercise 4b



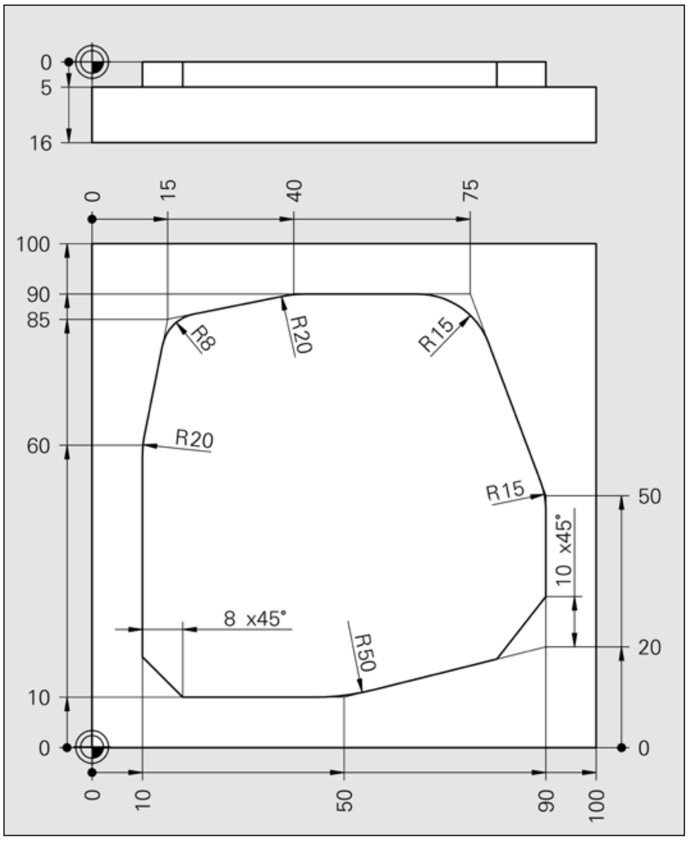
Exercise 4c



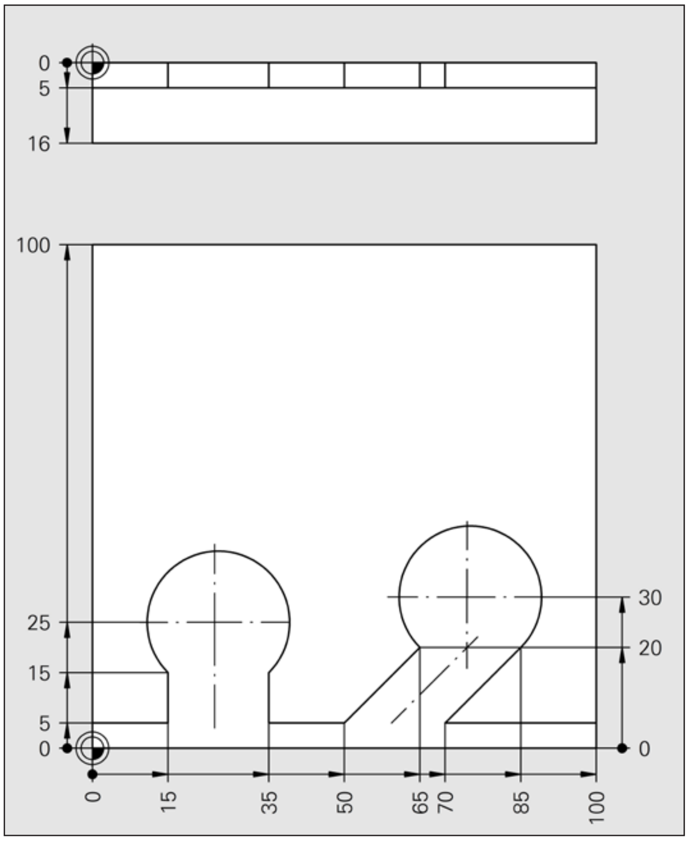
Exercise 4d



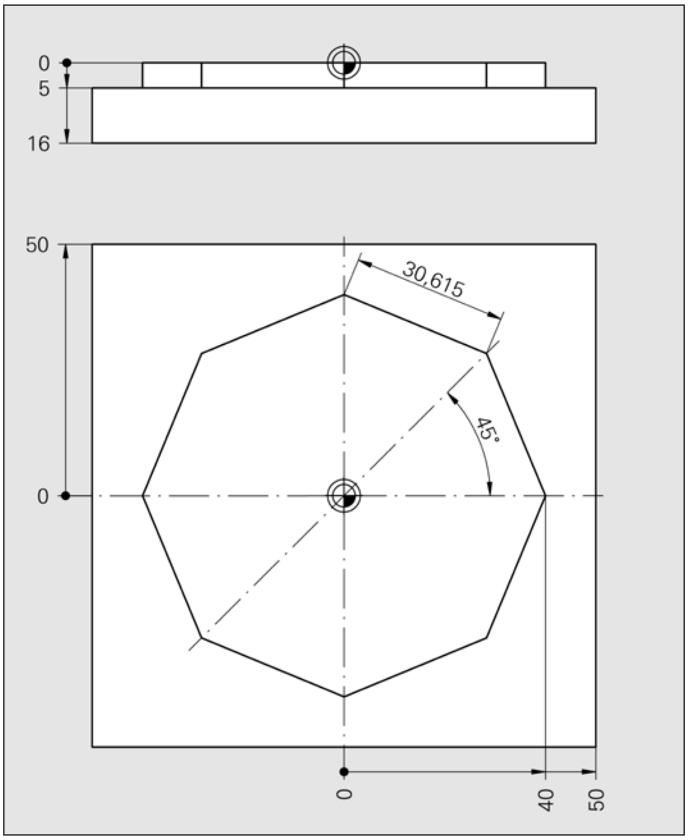
Exercise 5a



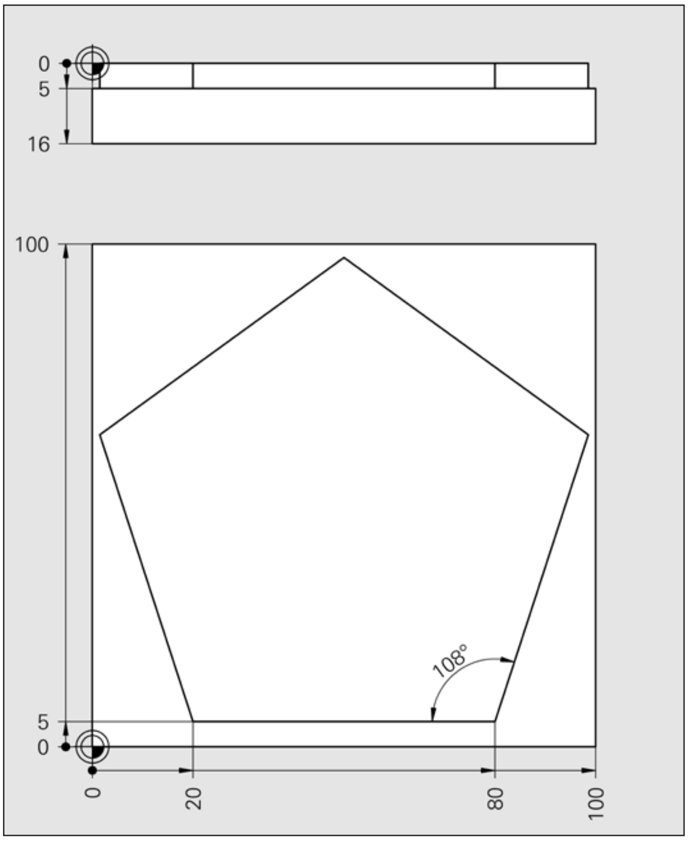
Exercise 5b



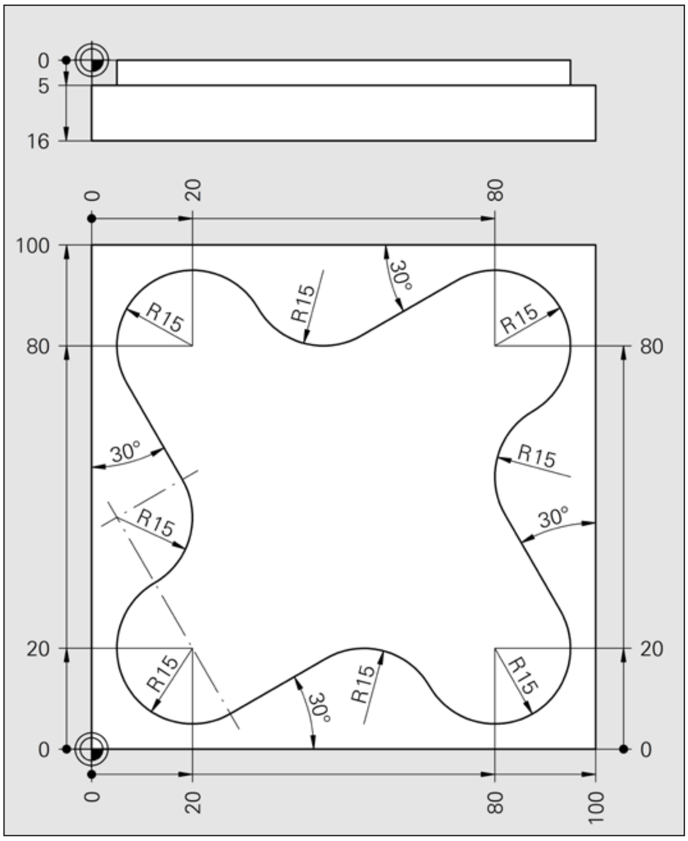
Exercise 5c



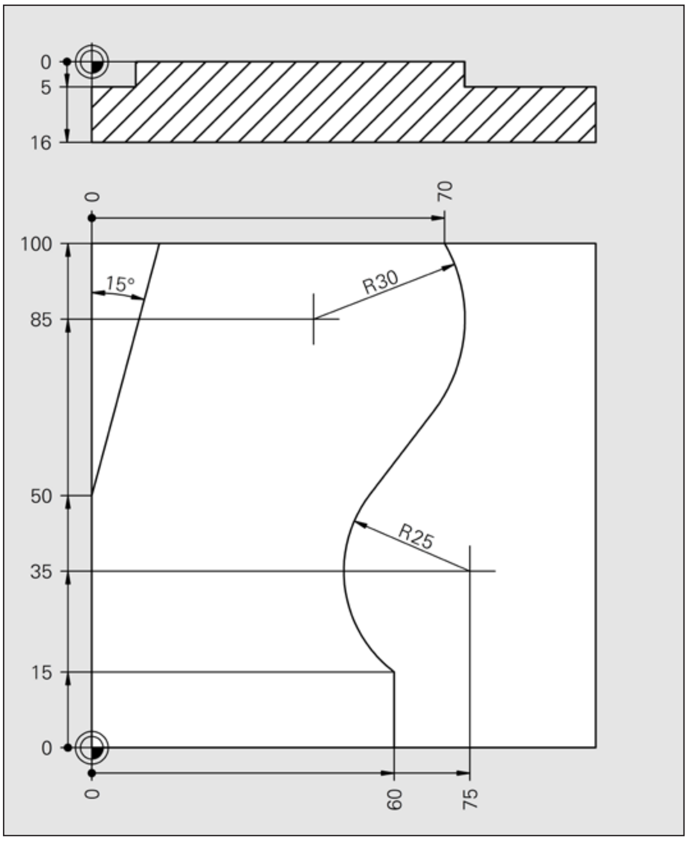
Exercise 5d



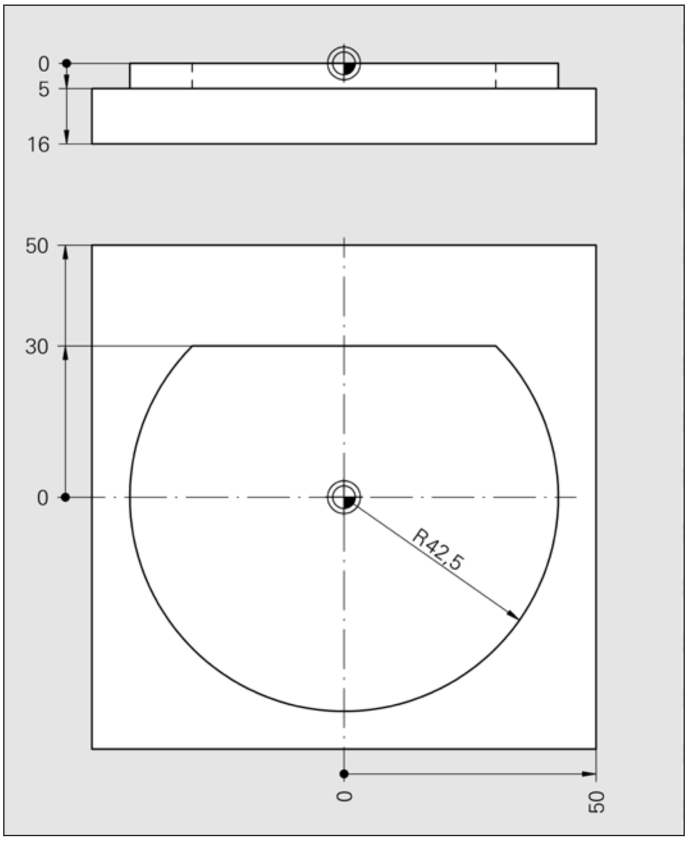
Exercise 6a



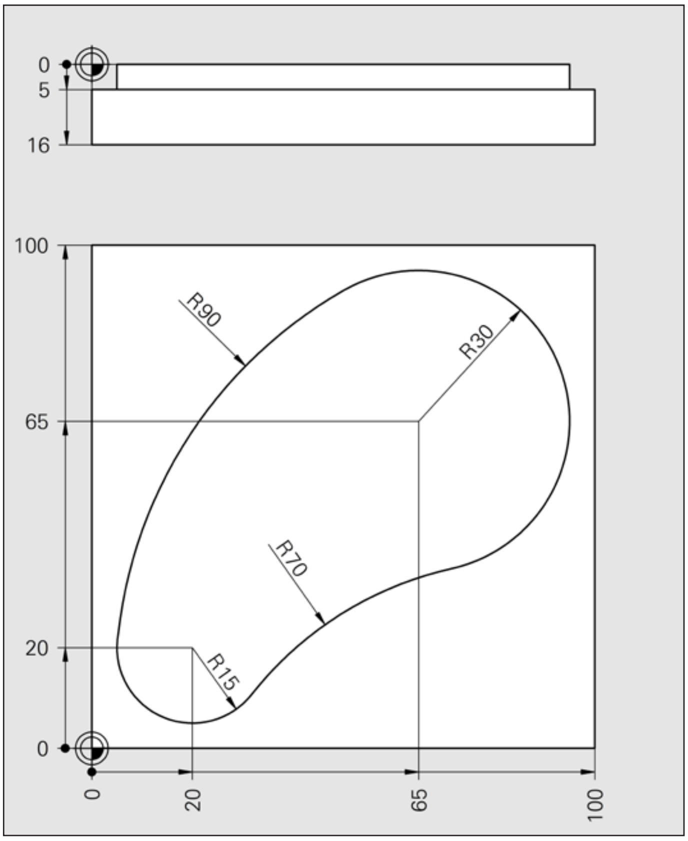
Exercise 6b



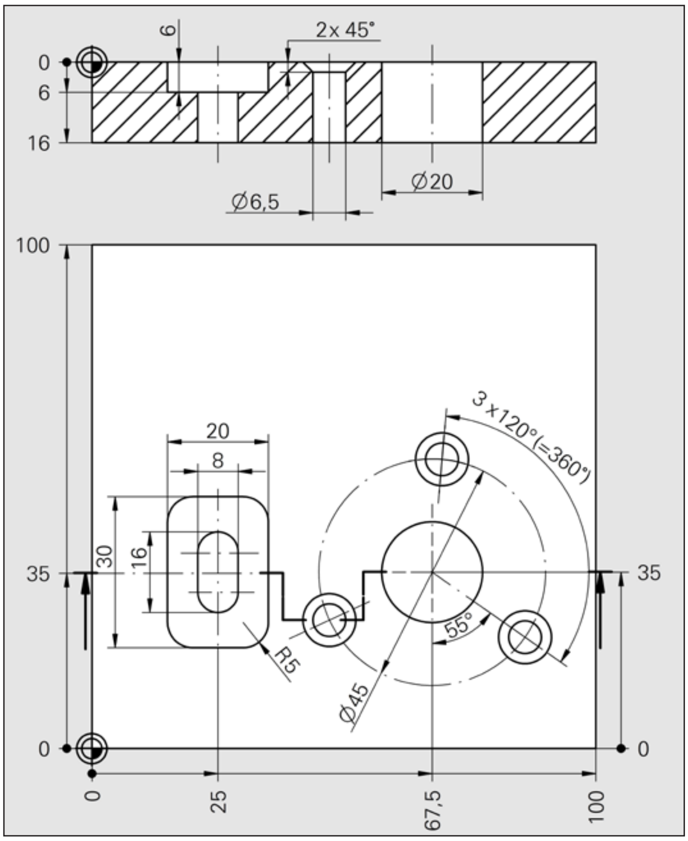
Exercise 6c



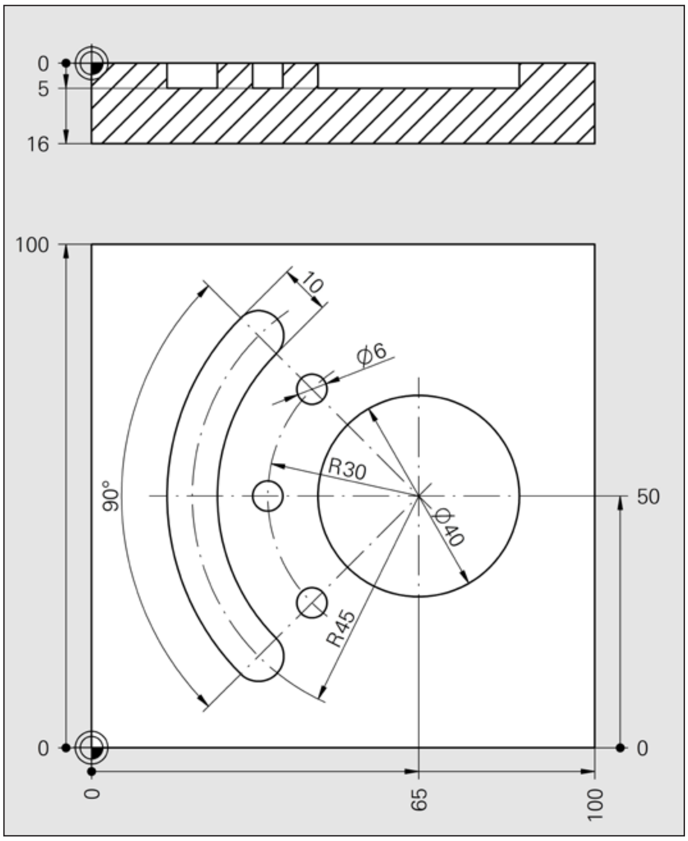
Exercise 6d



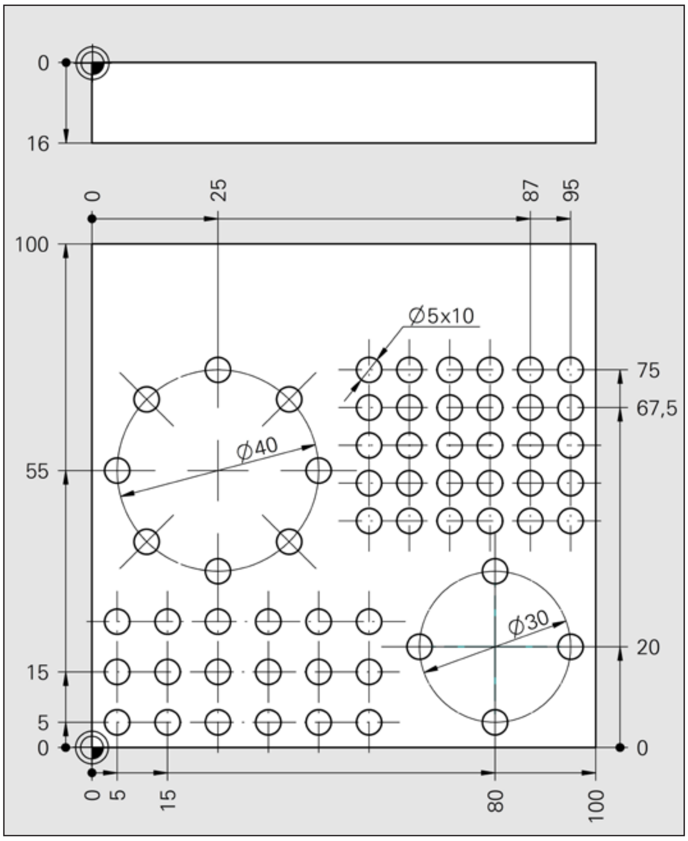
Exercise 7a



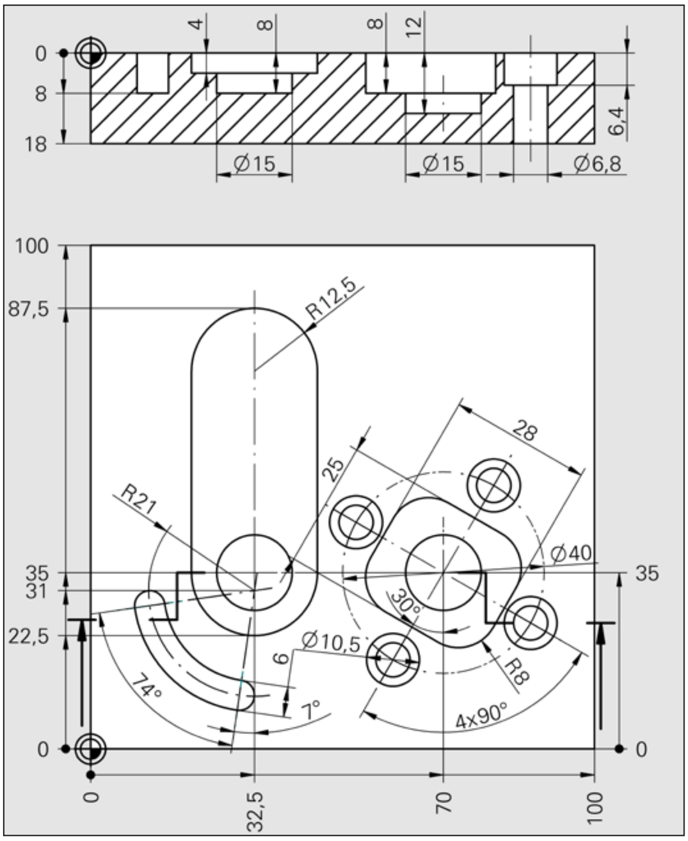
Exercise 7b



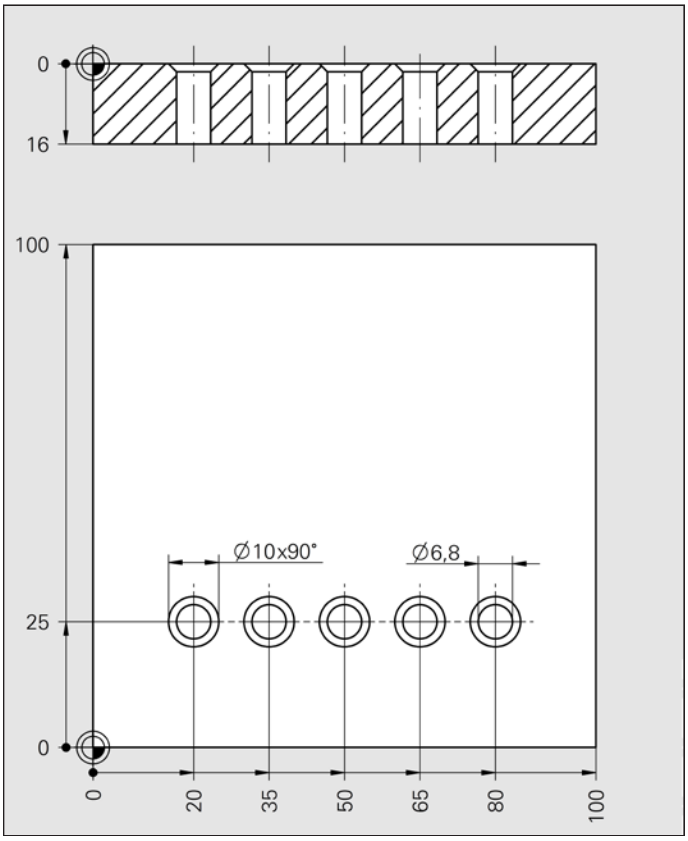
Exercise 7c



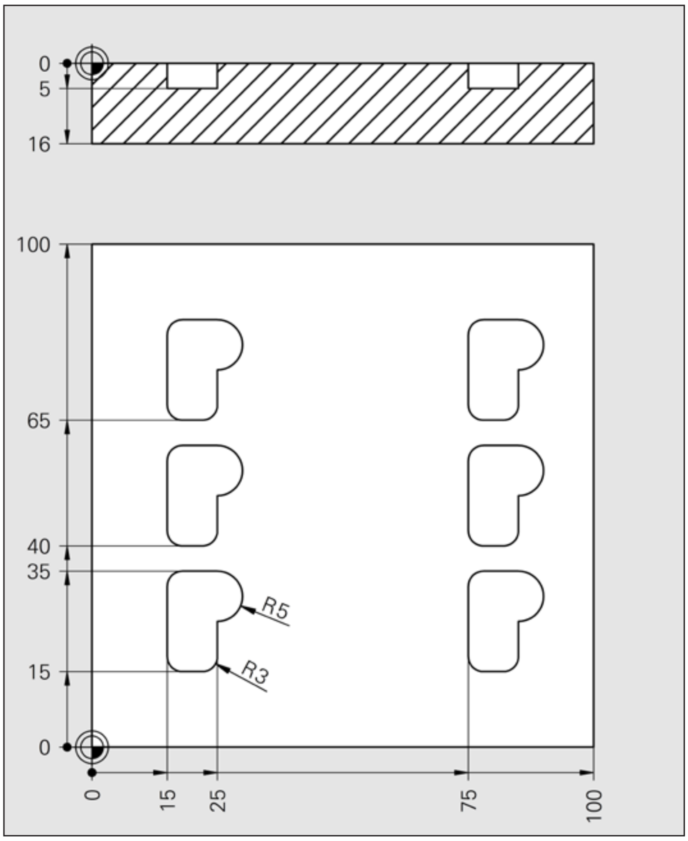
Exercise 7d



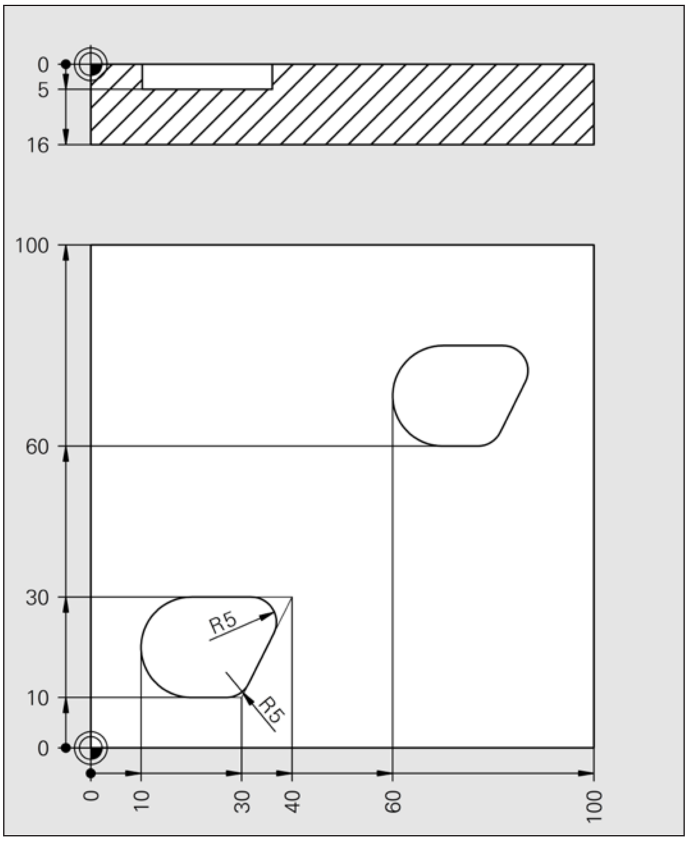
Exercise 8a



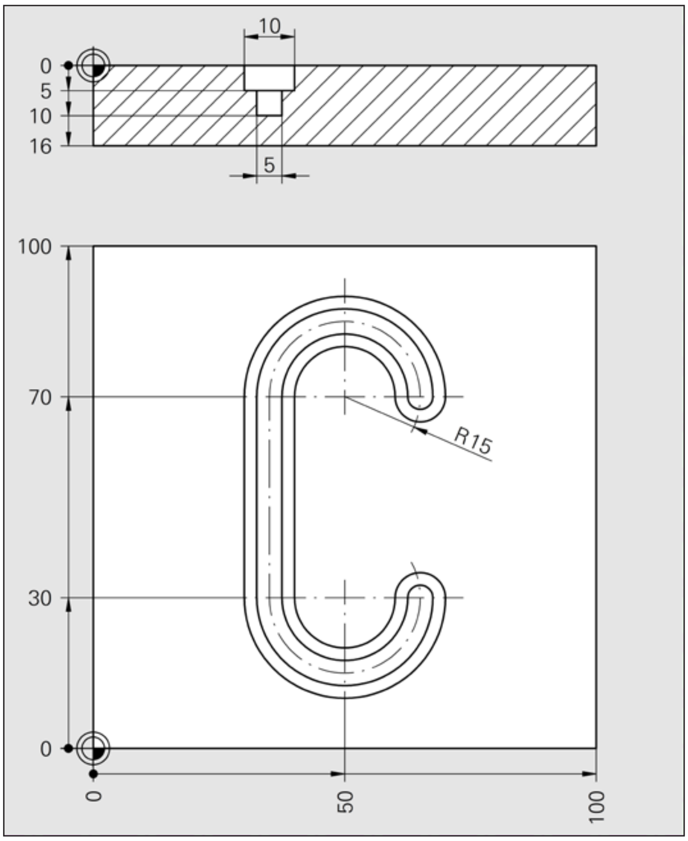
Exercise 8b



Exercise 8c



Exercise 8d



HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Deutschland
☏ +49 8669 31-0
FAX +49 8669 38609
E-Mail: info@heidenhain.de

Technische Schulung/Technical Training
☏ +49 8669 31-2293 oder 31-1695
FAX +49 8669 31-1999
E-Mail: service.training@heidenhain.de

www.heidenhain.de

809611-21 · Ver01 · Printed in Germany · 6/2012 · F&W